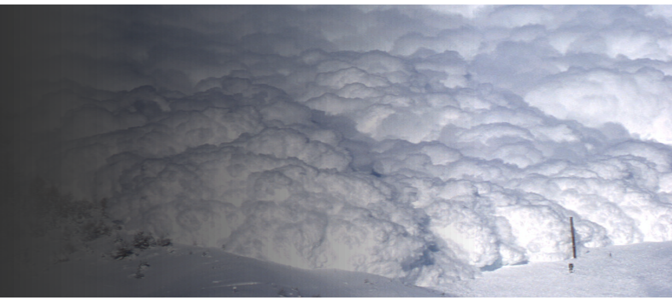
RAMMS

rapid mass movements simulation



A numerical model for snow avalanches in research and practice

User Manual v1.5
Avalanche

WSL Institut für Schnee- und Lawinenforschung SLF
WSL Institut pour l'étude de la neige et des avalanches SLF
WSL Instituto per lo studio della neve e delle valange SLF
WSL Institute for Snow and Avalanche Research SLF

Title picture: Vallée de la Sionne, WSL

Contributors (alphabetical order)

Perry Bartelt
Yves Buehler
Marc Christen
Yolanda Deubelbeiss
Maren Salz
Maike Schneider
Lina Schumacher

Manuscript update
March 7, 2013

Contents

1 Introduction

In the field of natural hazards there is an increasing need for process models to help understand the motion of geophysical mass movements. These models allow engineers to predict the speed and mass of hazardous movements in complex terrain. Such models are especially helpful when proposing mitigation measures, such as avalanche dams or snow sheds. Hazard mapping is an especially important application in Switzerland and other mountainous countries. An accurate prediction of runout distances, flow velocities and impact pressures in general three-dimensional terrain is the driving motivation for the development of dynamical mass movement models. Although helpful and well-liked by users, one-dimensional models such as AVAL-1D require that the primary flow direction and flow width must be defined by the user in advance. This is often difficult, especially in open terrain, or in terrain consisting of several possible flow channels. Furthermore, flow interaction with catching and deflecting dams cannot be accurately modeled using one-dimensional simulation codes.

RAMMS (Rapid Mass Movements Simulation) is a two-dimensional, state-of-the-art numerical simulation model to calculate the motion of geophysical mass movements (snow avalanches, rockslides, debris flows, shallow landslides) from initiation to runout in three-dimensional terrain. It was designed to be used in practice by hazard engineers who need solutions to real, everyday problems. It is coupled with a user-friendly visualization tool that allows them to easily access, display and analyze simulation results. New constitutive models have been developed and implemented in RAMMS, thanks to calibration and verification at full scale tests at sites such as Vallée de la Sionne. These models allow the application of RAMMS to solve both large, extreme avalanche events as well as smaller mass movements such as hillslope debris flows and shallow landslides.

RAMMS was developed by the RAMMS program team at the WSL Institute for Snow and Avalanche Research SLF. This manual describes the features of the RAMMS program - allowing beginners to get started quickly as well as serving as a reference to expert users.

The RAMMS web page `http://ramms.slf.ch` provides useful information such as a moderated discussion forum, frequently asked questions (FAQ) or recent software updates. Please visit this web page frequently to stay up to date!

DISCLAIMER

RAMMS is intended to be used as a tool to support experienced users. The interpretation of the simulation results has to be done by an avalanche expert who is familiar with the local as well as with the topographic and meteorological situation of the investigation area. In no event shall WSL/SLF be liable for any damage or lost profits arising, directly or indirectly, from the use of RAMMS. Swiss law applies. Court of jurisdiction is Davos. If you encounter problems, please contact *ramms@slf.ch*.

2 Learning by doing

This manual provides an overview of RAMMS. Exercises exemplify different steps in setting up and running a RAMMS simulation especially in **Chapter 4: Working with RAMMS**. However, to get the most from the manual, we suggest reading it through while simultaneously having the RAMMS program open, learning by doing. We assume RAMMS users to have a basic level of familiarity with windows-based programs, commands and general computer terminology. We do not describe the basics of windows management (such as resizing or minimizing). RAMMS windows, click options and input masks are similar to other windows-based programs and can be used, closed, reduced or resized in the same way.

3 Setup and first start

3.1 System requirements

We recommend the following minimum system requirements for running RAMMS:

- Operating System: Windows XP (32-bit) and Windows 7 (64-bit) (or Windows Virtual Machine VM)
- RAM (memory): 2GB (more recommended)
- CPU: Intel Pentium 1 GHz (dual core recommended)
- Harddisk: ca. 200 MB

3.2 Installation

Please download the RAMMS setup file *ramms_user_setup_64.zip* for Windows 7 and *ramms_user_setup.zip* for Windows XP from

http://ramms.slf.ch/ramms/downloads/ramms_user_setup_64.zip, respectively
http://ramms.slf.ch/ramms/downloads/ramms_user_setup.zip

Please do the following steps before starting to install RAMMS:

- Click on the path given above or copy the path to any browser. A window pops up and the automatic download of the file *ramms_user_setup_64.zip* or *ramms_user_setup.zip* start after clicking *yes*.
- You must have *Administrator privileges* on the target machine. If you do not have such privileges, the installer cannot modify the system configuration of the machine and the installation will fail. Note that you do not need Administrator privileges to run RAMMS afterwards.
- Read first, install afterwards! Please read the whole installation process once, before you begin the installation!

3.2.1 Installation procedure

Step 1: Welcome

Start the file *ramms1.5.01_user_setup_64.exe*. The welcome dialog introduces you to the English setup program and will guide you through the installation process. Click *Next* to continue.

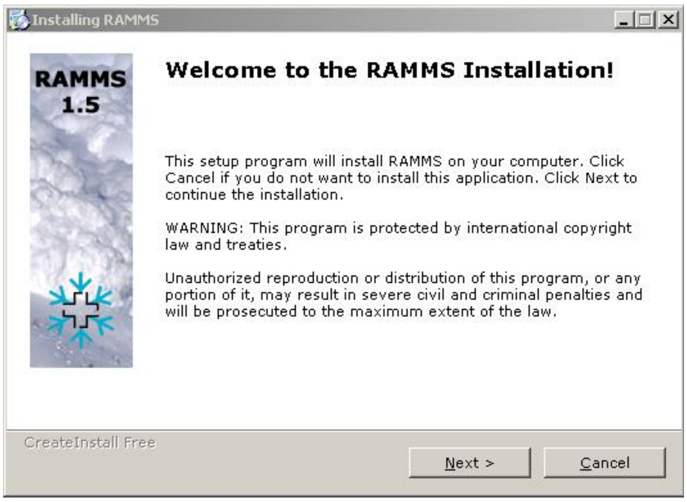


Figure 3.1: Installation - welcome dialog window.

Step 2: Readme

Short introduction to RAMMS. Click *Next* to continue.

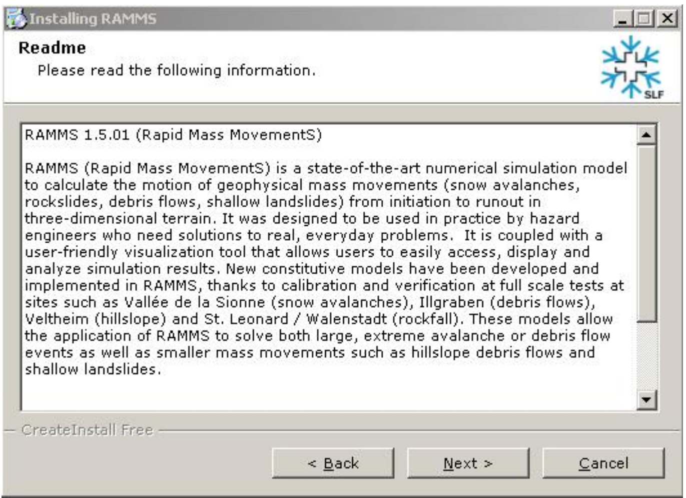


Figure 3.2: Installation - readme dialog window.

Step 3: Accepting the license agreement

Read the license agreement carefully and accept it by activating the check box in the lower left corner. If you do not accept the license agreement, you are not able to proceed with the installation. After accepting the license agreement, click *Next* to continue the installation.

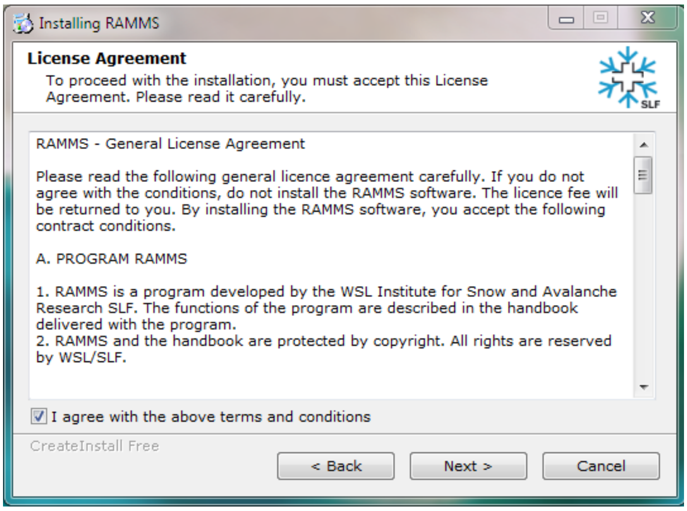


Figure 3.3: Installation - license agreement dialog window.

Step 4: Select destination directory

Choose your destination directory. Simultaneously this dialog shows the amount of space available on your hard disk and required for the installation. Click *Next* to start the installation process.

Figure 3.4: Installation - destination directory dialog window.

Step 5: Installing the files

RAMMS is copying the files to the destination location and showing the installation progress.

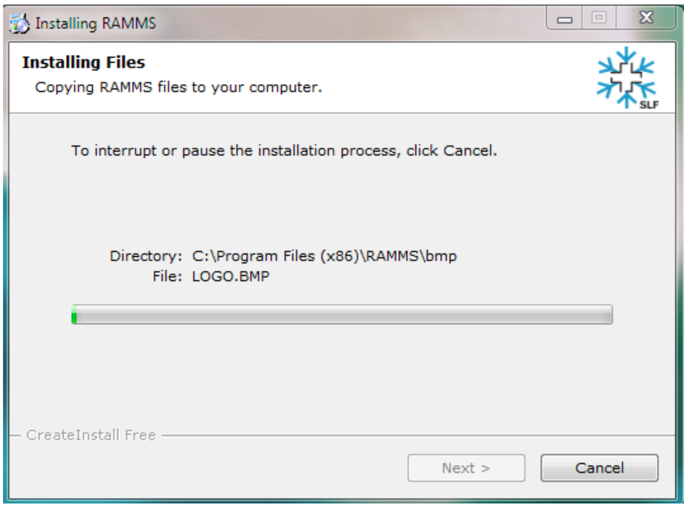


Figure 3.5: Installation - installing files dialog window.

Step 6: Finished installing the files

RAMMS finished copying the files. Click *Next* to finish the installation process.

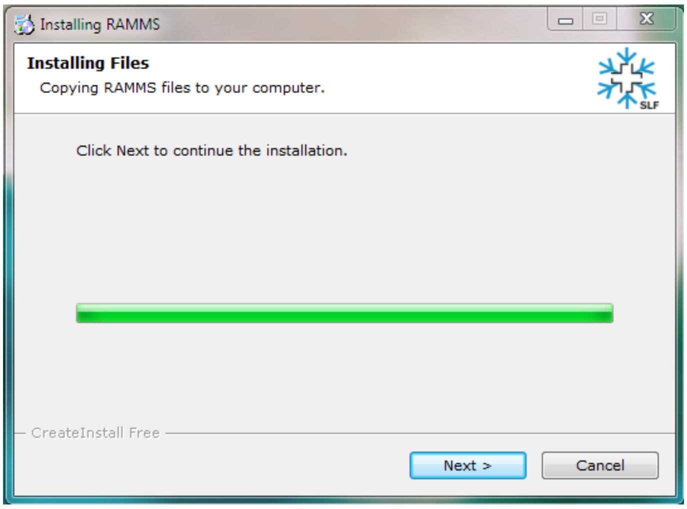


Figure 3.6: Installation - finished installing files dialog window.

Step 7: RAMMS installation finished!

RAMMS successfully finished the installation. Click *Finish*.

Figure 3.7: Installation - finished installation dialog window.

Step 8: Welcome to IDL Visual Studio Merge Modules

To ensure all important system libraries are installed on your target machine follow the instructions below:

The welcome dialog introduces you to the English setup program and will guide you through the installation process of the IDL Visual Studio Merge Modules. Click *Next* to continue.

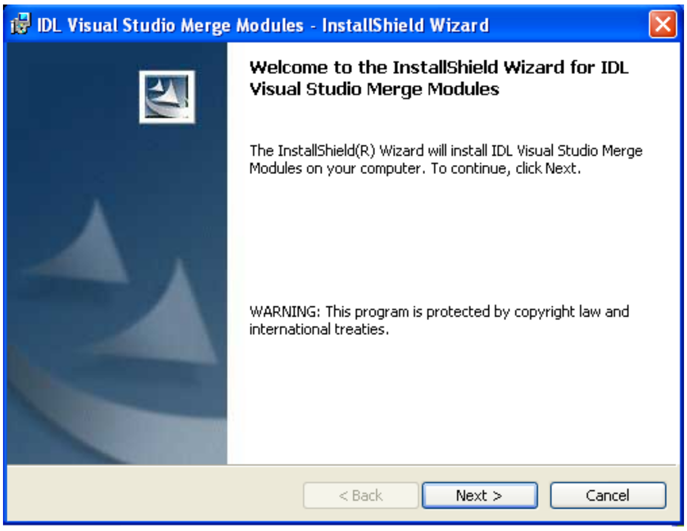


Figure 3.8: IDL Visual Studio Merge Modules - welcome dialog window.

Step 9: Ready to install the program

Click *Next* to continue.

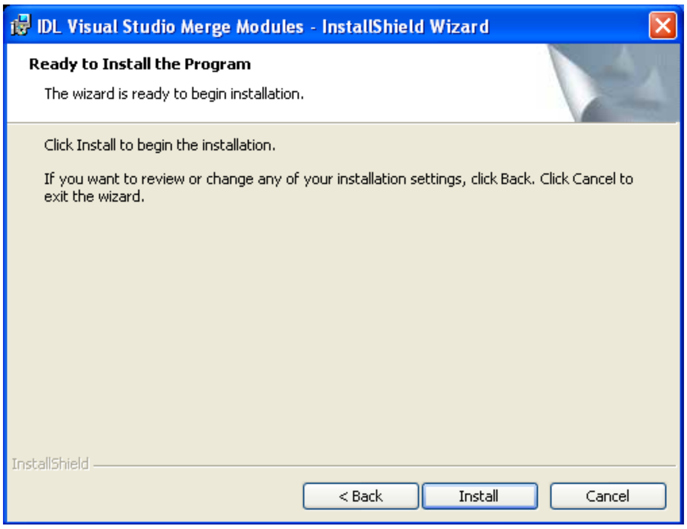


Figure 3.9: IDL Visual Studio Merge Modules - ready to install the program.

Step 10: Installing IDL Visual Studio Merge Modules

The wizard is installing the files. Please wait until it is finished.

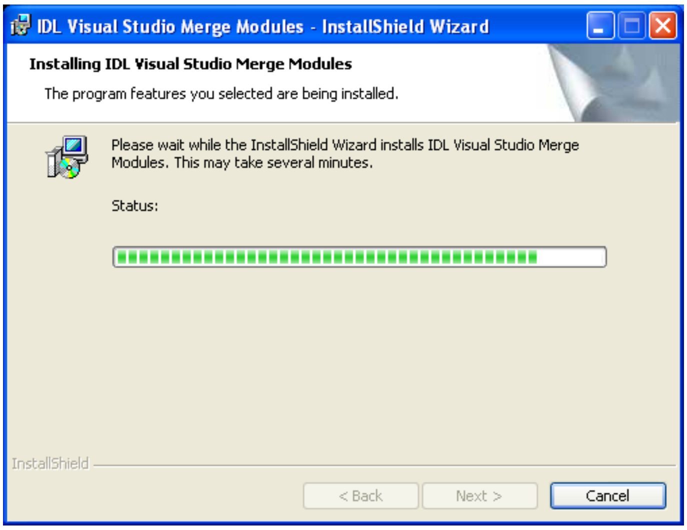


Figure 3.10: IDL Visual Studio Merge Modules - installing...

Step 11: InstallShield Wizard Completed

The wizard completed the installation. Click *Finish*.

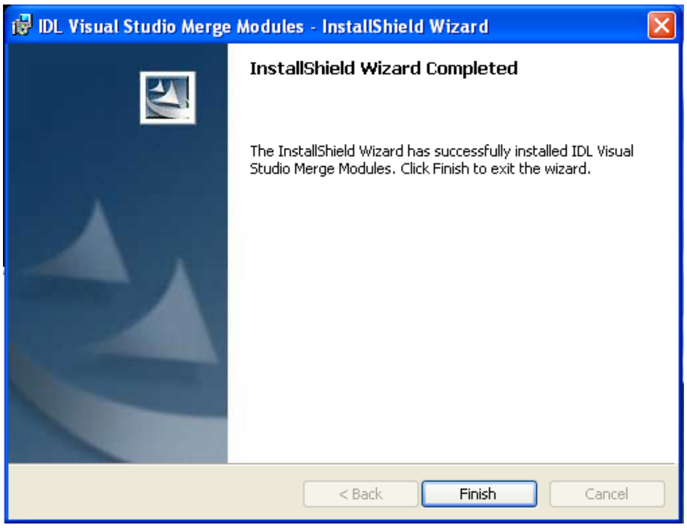


Figure 3.11: Installation - destination directory dialog window.

After having successfully installed RAMMS and the necessary files on your personal computer, you will notice the RAMMS icon on your desktop (for all users):

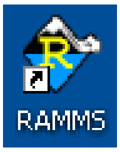


Figure 3.12: RAMMS icon.

Additionally, a new application folder is created in *Start* → *Programs* (for all users):

- RAMMS → Run RAMMS
- RAMMS → Uninstall RAMMS

Figure 3.13: RAMMS program group.

3.3 Licensing Methods

Access to RAMMS is controlled by a personal use license. Personal use licenses are time limited licenses tied to a single personal computer. This method of licensing requires a machine's unique host ID to be incorporated into a license request file. After the license request file is sent to WSL/SLF, you will receive a license key. Entering the license key on a personal computer enables full RAMMS functionality for the specific personal computer. For more information please visit http://ramms.slf.ch.

3.4 First Start

Double-click the icon or use *Start* → *Programs* → *RAMMS* → *Run RAMMS* to start RAMMS for the first time. Whenever you start RAMMS, the splash screen below will pop up:

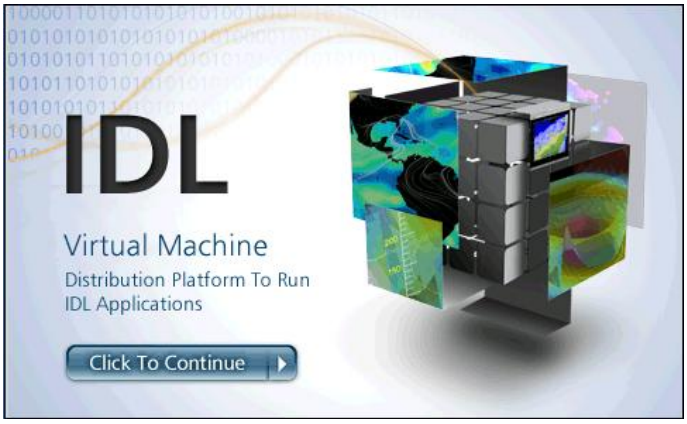


Figure 3.14: RAMMS start window.

Click on the image. It will disappear and RAMMS will start up. The following dialog window appears (Fig. 3.15 RAMMS - Licensing):

Figure 3.15: RAMMS licensing window.

3.4.1 Personal license request file

Click the button to create your personal license request file. In Fig. 3.16 enter your full name and the name of your company.

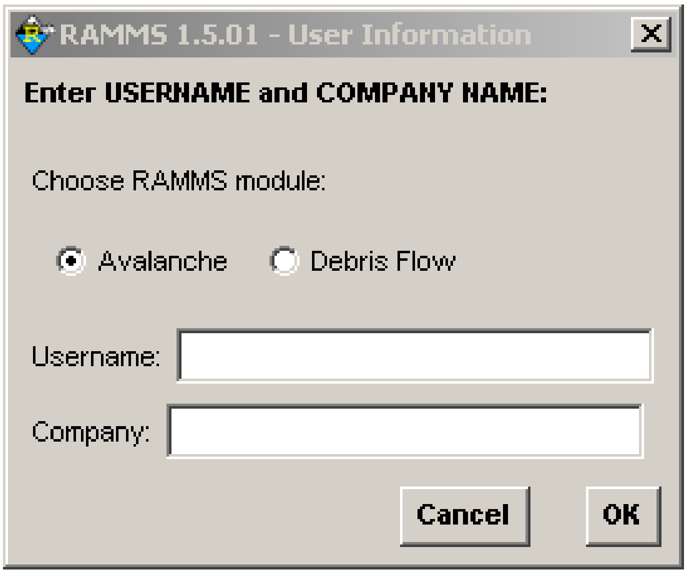


Figure 3.16: Enter user name and company name.

In the next dialog window, choose the destination directory of your personal license request file and save it to your target machine. Your personal license request file should look similar to Fig. 3.17.

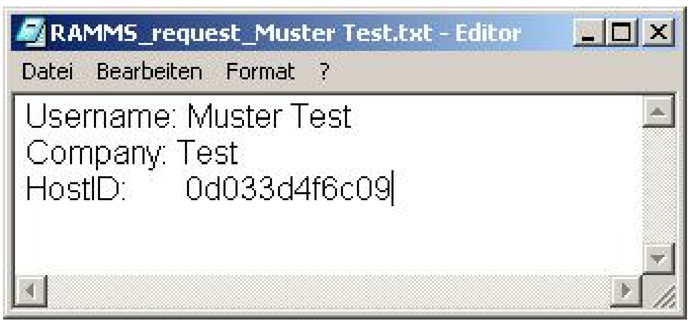


Figure 3.17: Personal license request file *RAMMS_request_Muster Test.txt*

3.4.2 Get the personal license key

You find an order form on the RAMMS web page (*Order Form* or *Demo Order Form*) at http://ramms.slf.ch. Fill in all your personal information, choose license period, license typ and number of licenses you wish to order, attach your personal license request file(s), accept the license agreement and click *Submit Order*.

An order confirmation email is sent to your email address. We then process your order and send you an invoice. As soon as we received your payment, we will send you your personal license key. Your personal license key is named similar to *RAMMS_license_Muster Test.txt*. Open the file in a text-editor. It should look similar to Fig. 3.18.

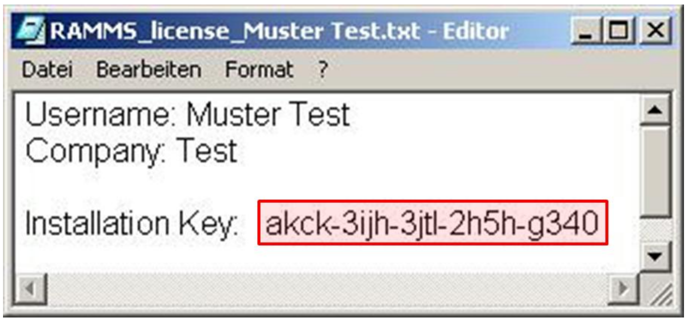


Figure 3.18: Personal license key file *RAMMS_license_Muster Test.txt*

Now, restart RAMMS (as explained before). Again, the pop-up window (Fig. 3.14) and then the dialog window of Fig. 3.15 appears (RAMMS - Licensing). Copy the license key (in this example: *akck-3ijh-3jtl-2h5h-g340*) and paste it at *license key:* (see Figure 3.15). Notice that there might be a prefix AVALANCHE. This prefix is part of the license key and has to be inserted as well! If RAMMS accepts your installation key, you successfully finished the installation.

3.5 Update

When you start RAMMS it will automatically check for updates on the internet. This could lead to an error message, if your firewall blocks the executable *idlrt.exe* (this file starts the IDL-Virtual Machine you need to run RAMMS). Please unblock this file for your firewall. You can also disable the AutoWebUpdate-function by unchecking *Help* → *Advanced...* → *AutoWebUpdate*. The WebUpdate-function can still be done manually under *Help* → *Update...* → *Web Update* or go to *Help* → *Update...* → *get manually* and download the file to your local folder.

3.6 Preferences

Before starting to work with RAMMS, be sure to set your RAMMS preferences and place the necessary DEM (Digital Elevation Model) files as well as the FOREST files, MAPS and georeferenced IMAGERY you wish to use in the appropriate folders defined in the preferences, see Figs. 3.19 and 3.20.
Use *Track* → *Preferences* to open the RAMMS preferences window or click the button.
For resetting the general preferences use *Help* → *Advanced...* → *Reset General Preferences*.

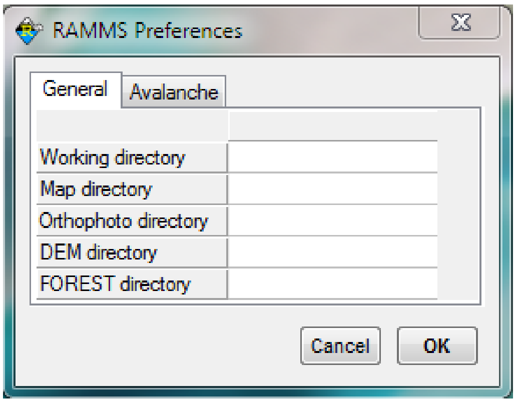


Figure 3.19: General tab of RAMMS preferences.

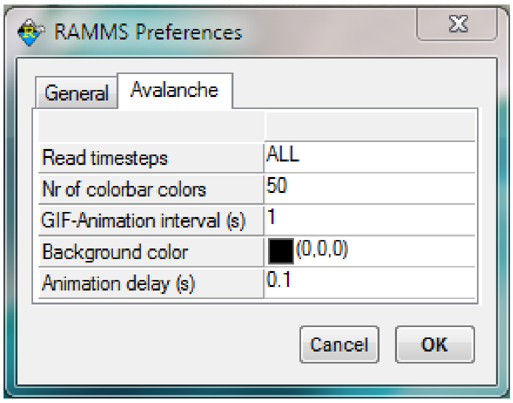


Figure 3.20: Avalanche tab of RAMMS preferences.

General tab

Setting	Purpose
Working directory	Set your working directory. **VERY IMPORTANT: Do NOT use BLANKS** in the working directory path!
Map directory	Set the folder where you place your georeferenced digital maps (consists of a TIFF-file and a corresponding tfw-file (world-file).
Image directory	Set the folder where you place your digital georeferenced orthophotos (aerial picture, consists of a TIFF-file and a corresponding tfw-file (world-file).
DEM directory	Set the folder where you place the **D**igital **E**levation **M**odels (format: ASCII grid, see 4.2.1 on page 20)
FOREST directory	Set the folder where you place your forest-files (formats: ASCII grid or polygon shapefile).

Avalanche tab

Setting	Purpose
Read timesteps	Choose between reading ALL or only 1 timestep. Default is reading ALL timesteps.
Nr of colorbar colors	Set default nr of colorbar colors.
GIF-Animation interval (s)	Set interval for GIF-Animation images.
Background color	Set background color (greyscale between 0:black and 255:white).
Animation delay (s)	Set animation delay to decelerate the animation speed.

The following exercise *"Working directory"* shows how to choose a new working directory. All further settings can be changed in a similar manner. The settings are saved, until they are changed again manually.

Exercise 3.6.a: Working directory

Choosing the right working directory is very useful and saves a lot of time, searching for files and folders.

VERY IMPORTANT: Do NOT use blanks or special characters in the path names!

- Click (or use **Track** → **Preferences**) to open the RAMMS preferences window.
- Click into the field *Working directory*, then the appearing arrow and **Edit...**
  A window pops up where you can choose your new working directory. Click **OK** in both windows.

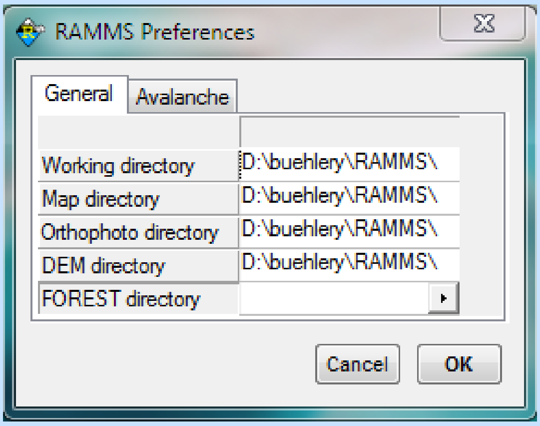


Figure 3.21: RAMMS preferences.

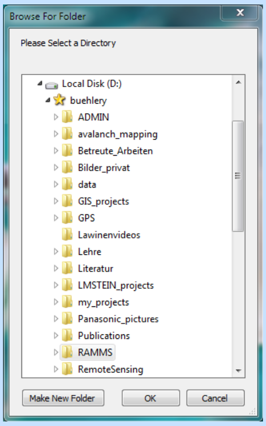


Figure 3.22: Browse for the correct folder.

End of Example 3.6.a

4 Working with RAMMS

All topographic base maps and aerial images shown in this manual are reproduced ©2012 swisstopo (JD100007).

4.1 Preparations

To successfully start a new RAMMS project, a few important preparations are necessary. Topographic input data (ascii format), project boundary coordinates and georeferenced maps or remote sensing imagery should be prepared in advance (.tif format and .tfw file, maps and imagery are not mandatory, but nice to have). Georeferenced datasets have to be in a Cartesian coordinate system (e.g. Swiss CH1903 LV03). Polar coordinate systems (e.g. WGS84 Long Lat) are not supported. For more information about specific national coordinate systems please contact the national topographic agency in your country.

4.1.1 Project and Scenarios

A project is defined for a region of interest. Within a project, one or more scenarios can be specified and analyzed. For every scenario, a calculation can be executed. A project consists therefore of different scenarios (input files) with different input parameter files (release and friction files). The basic topographic input data is the same for every scenario. If you want to change the topographic input data (e.g. change the input DEM resolution or the project boundary coordinates) you have to create a new project. Other input parameters (like friction parameters, release areas, calculation domain, calculation grid resolution, end time, time step etc.) can be changed for every scenario.

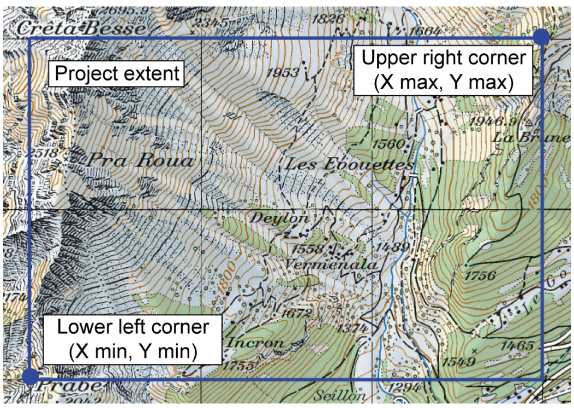


Figure 4.1: The same project extent (area of interest) can be used to calculate different scenarios with different input parameters.

4.2 Model input data

There are different kind of data to be provided to successfully perform a calculation with RAMMS. Topographic data, definition of release area and release volume as well as information about friction parameters are mandatory.

4.2.1 Topographic data - Digital Elevation Model (DEM)

The topographic data is the most important input requirement. The simulation results depend strongly on the resolution and accuracy of the topographic input data. Before you start a simulation make sure all important terrain features are represented in the input DEM. RAMMS is able to process the following topographic data:

- ESRI ASCII grid (Fig. 4.2)
- ASCII X,Y,Z single space data (Fig. 4.3)

These data types are also available e.g. from `www.swisstopo.ch`. Because RAMMS needs the topographic data as an ESRI ASCII grid, ASCII X,Y,Z data can be converted within RAMMS into an ESRI ASCII grid. At this stage no other data types are processable. The user must therefore prepare the topographic data according to this limitation. The header of an ESRI ASCII grid must contain the information shown in Fig. 4.2

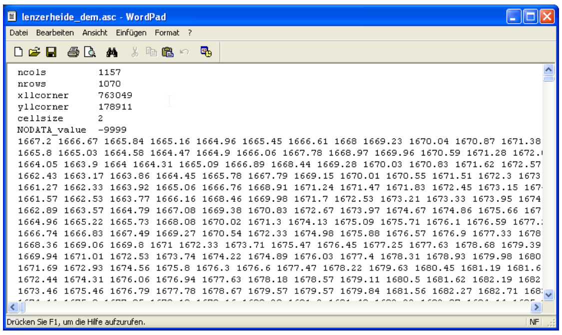


Figure 4.2: Example ESRI ASCII grid.

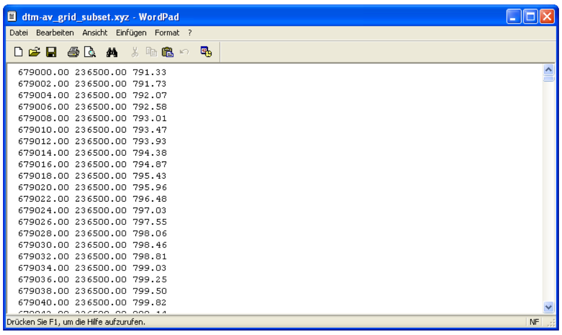


Figure 4.3: Example ASCII X,Y,Z single space data.

Conversion into ESRI ASCII grid

An ESRI ASCII grid can be created in ArcGIS with the function *ArcToolbox* → *Conversion Tools* → *From Raster* → *Raster to ASCII*.

It is possible to import ASCII X,Y,Z single space data and convert the data into an ESRI ASCII grid (using *Track* → *New...* → *Convert XYZ to ASCII grid*).

4.2.2 Release information

The definition of release areas and release heights have a very strong impact on the results of RAMMS simulations. Therefore we recommend to use reference information such as photography, GPS measurements or filed maps to draw release areas. This should be done by people with experience concerning the topographic and meteorological situation of the investigation area.

Users have to draw their own release polygon shapefiles, see section 4.5.1 on page 36. All release informations are saved as polygon shapefiles and can be easily imported in GIS-Software (e.g. ArcGIS). Shapefiles created in e.g. ArcGIS can be imported into RAMMS by using *GIS → Convert Polygon Shapefile... → to RAMMS Release Shapefile*.

4.2.3 Friction information

RAMMS employs a Voellmy-fluid friction model, which is based on the Voellmy-Salm approach (we refere to Salm et al. 1990 [3] and Salm 1993 [4]).

Physical friction model

The physical model of RAMMS::Avalanche uses the Voellmy friction law. This model divides the frictional resistance into two parts: a dry-Coulomb type friction (coefficient μ) that scales with the normal stress and a velocity-squared drag or viscous-turbulent friction (coefficient ξ). The frictional resistance S (Pa) is then

$$S = \mu \rho H g cos(\phi) + \frac{\rho g U^2}{\xi}, \tag{4.1}$$

where ρ is the density, g the gravitational acceleration, ϕ the slope angle, H the flow height and U the flow velocity. The normal stress on the running surface, $\rho H g cos(\phi)$, can be summarized in a single parameter N. The Voellmy model accounts for the resistance of the solid phase (μ is sometimes expressed as the tangent of the internal shear angle) and a viscous or turbulent fluid phase (ξ was introduced by Voellmy using hydrodynamic arguments). The friction coefficients are responsible for the behavior of the flow. μ dominates when the flow is close to stopping, ξ dominates when the flow is running quickly.

The Voellmy friction model has found wide application in the simulation of mass movements, especially snow avalanches. For modeling snow avalanches the Voellmy model has been in use in Switzerland for many years and a set of standard parameters is available.

Friction parameters μ and ξ

RAMMS::Avalanche offers a constant and a variable calculation mode. If a calculation is done with constant friction values, of course, no terrain undulations and forest areas are considered. Therefore we suggest to use the variable friction values if possible. An automatic RAMMS procedure classifies friction values (μ and ξ) based on topographic data analysis (slope angle, altitude and curvature), forest information and global parameters *return period* and *avalanche volume* (see Fig. 4.4).

μ and ξ values are saved as ASCII files (called MuXi-files) and can be easily imported in GIS-Software (e.g. ArcGIS), see section 4.2.3 on page 21.

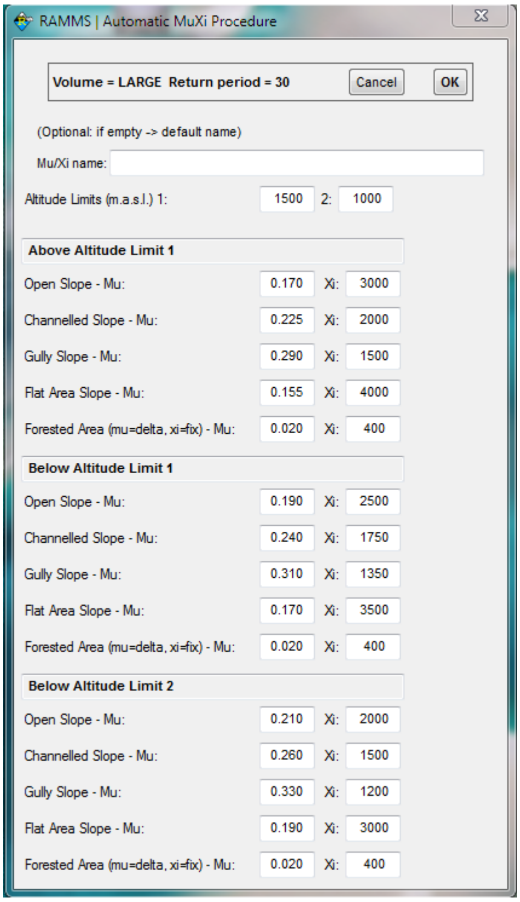


Figure 4.4: RAMMS | Automatic MuXi Procedure.

How to create a new MuXi-file is demonstrated in the exercise *"How to create a new MuXi-file"* on page 43.

4.2.4 Global parameters

The friction values μ and ξ strongly depend on the global parameters *return period* and *avalanche volume* (see the MuXi-table on page 89). Therefore an appropriate return period has to be definded and the avalanche volume has to be checked under *Input* $\rightarrow$ *Global Parameters* prior to create a new MuXi-file (see Fig. 4.5 and exercise *"How to create a new MuXi-file"* on page 43).

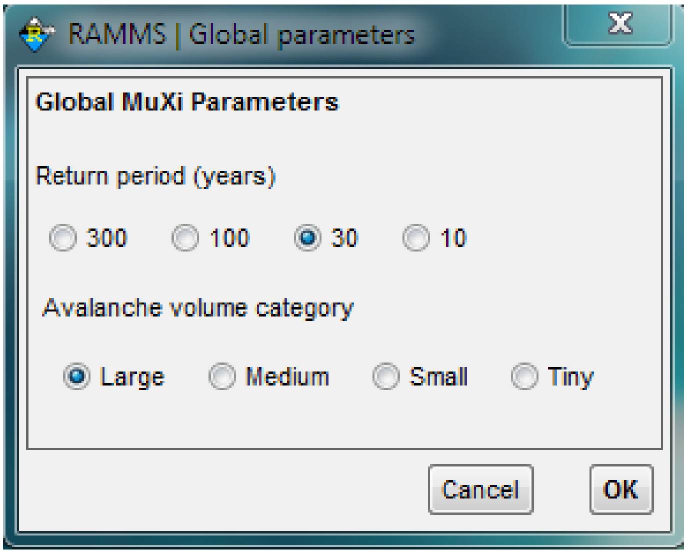


Figure 4.5: RAMMS global parameters.

4.2.5 Forest information

Forest information is not required for a successful simulation, but recommended, because the friction parameters strongly depend on forest information. Forest information can be provided as:

- ESRI ASCII grid (0: no forest, 1: forest)
- Polygon shapefile

If no such files are available, the user can draw a polygon shapefile in RAMMS and import it as forest information (see section 4.5.3 on page 41).

4.2.6 Calculation parameters

Calculation parameters such as output name, simulation grid resolution, end time, time step etc. can be changed interactively in the RAMMS *Run Simulation Widget*.

4.3 Creating a project with the Project Wizard

A new project is created with the RAMMS Project Wizard, shown in the exercise below. The wizard consists of four steps:

Exercise 4.3.a: How to create a new project.

- Click  or **Track** → **New...** → **Project wizard** to open the RAMMS Project Wizard.
- The following window pops up:

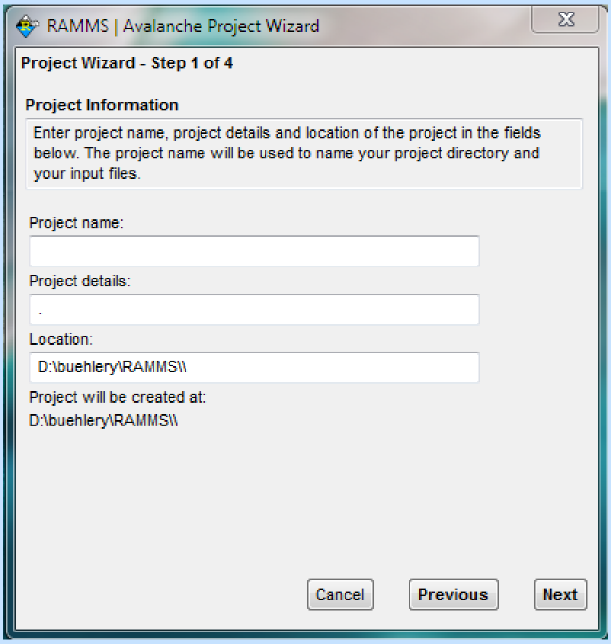


Figure 4.6: RAMMS Avalanche Project Wizard: Step 1 of 4.

Continuation of exercise 4.3.a: How to create a new project.

Step 1:

- Enter a project name (1).
- Add some project details (2).
- The project location (3) suggested is the current working directory. To change the location click into the location field. A second window appears and you can browse for a different folder (see figure below, VERY IMPORTANT: Do NOT use BLANKS or special characters in the project location path!).
- Click **Next** (4).

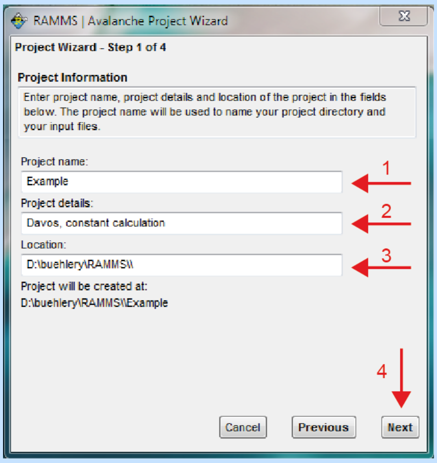


Figure 4.7: Step 1 of the RAMMS project wizard: Project information.

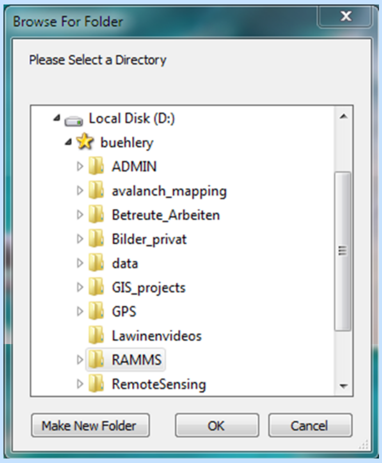


Figure 4.8: Window to browse for a new project location.

Step 2:

- Locate your DEM- and FOREST-file in the folder set in the RAMMS preferences. Click into the corresponding fields to browse for the appropriate files (1).
- If you don't want to use a FOREST-file, select "Do NOT use forest information" (2).
- Click **Next** (3).

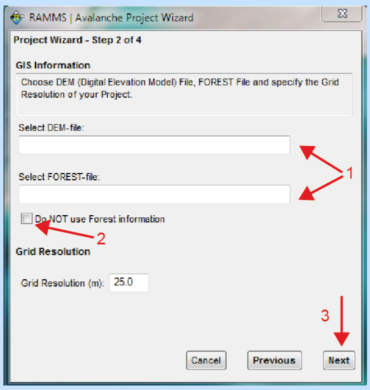


Figure 4.9: Step 2 of the RAMMS project wizard: GIS information.

Continuation of exercise 4.3.a: How to create a new project.

Step 3:

- Enter the X- and Y-coordinates of the lower left and upper right corner of your project area, using the Swiss Coordinate System CH1903 LV03 (or another cartesian coordinate system), as it is shown below for the Vallée de la Sionne area.

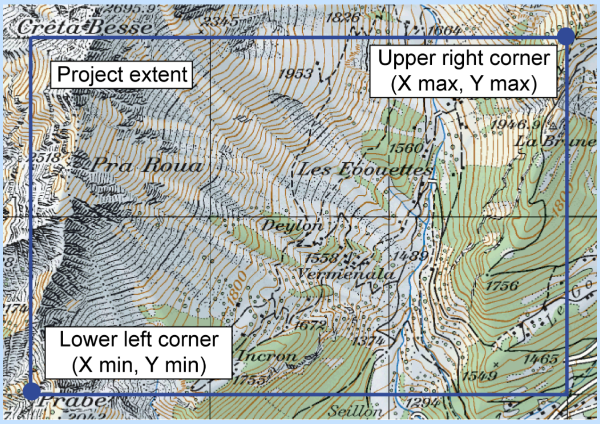


Figure 4.10: Project coordinates: lower left and upper right corner of project area.

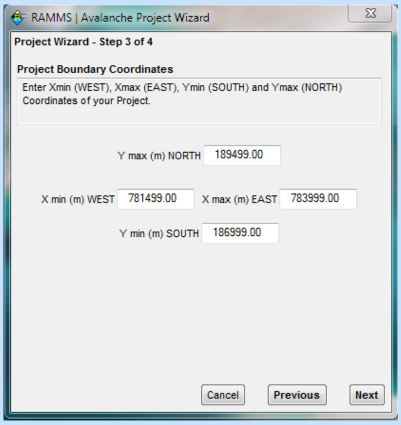


Figure 4.11: Step 3 of the RAMMS project wizard: Project boundary coordinates.

Step 4:

- Check the project summary, especially if a DEM- and FOREST-file was found.
- If several matching tif files exist, RAMMS shows a list with all these files.
- To make changes click **Previous**, to create the project click **Create Project**.

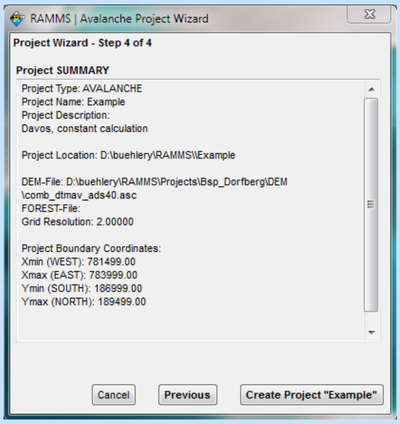


Figure 4.12: Step 4 of the RAMMS project wizard: Project summary.

Project creation:

- The creation process can take a while. Different status bars will pop up and show the progress of the project creation process.

End of exercise 4.3.a

The following files will be created in the project-folder

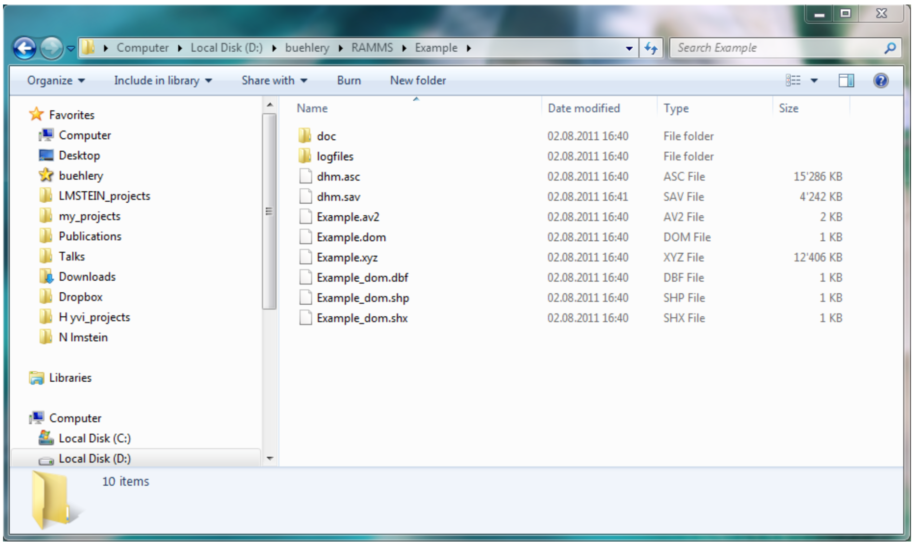


Figure 4.13: Created project files.

File / Folder	Purpose
doc (folder)	Folder containing input and ouput Log files
logfiles (folder)	Project creation and calculation log files
dhm.asc	ASCII grid with altitude values
dhm.sav	Height information used in RAMMS
_.av2	Input file
_.dom	Calculation domain ASCII file
_dom.shp	Calculation domain shapefile
_dom.shx	Calculation domain shapefile
_dom.dbf	Calculation domain shapefile
_.xyz	Topographic data used in RAMMS

4.4 Working with the interface

Once the project is created, there are several useful tools which can be helpful when working with RAMMS. They are explained in the excercises below.

4.4.1 Moving, resizing, rotating, viewing

Exercise 4.4.a: Moving and resizing the model

a) Terrain model has a dimension of 100% or smaller:

- By clicking on the "arrow" , the model can be moved and resized.

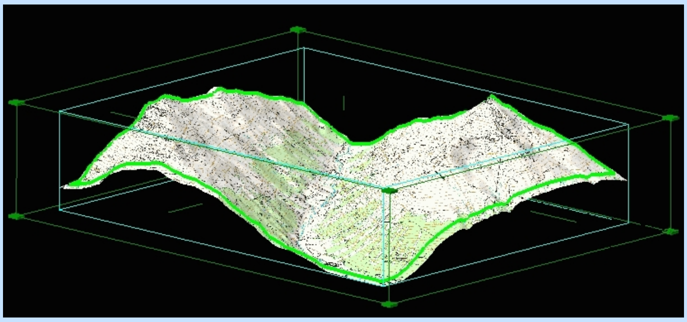
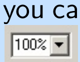


Figure 4.14: "Active" project with lines and corners for resizing.

- To move the model without changing size or aspect ratio, move to the model and check if the cursor turns to . Then click and hold the left mouse button and drag the model to the desired position.

- To resize the model without changing the aspect ratio, use the mouse wheel to zoom in our out. Alternatively, you can resize the model by changing the percentage value in the horizontal toolbar .

b) Terrain model has a dimension > 100%:

- All steps explained above are still possible.
- In addition to this, the white hand right next to the rotation button becomes active as well. After clicking on this so-called "view pan" button , it is also possible to move the model.

End of exercise 4.4.a

Exercise 4.4.b: Rotating the model

- After activating the rotation button, the model can be rotated along the rotation axis, by moving the cursor directly on one of the axis until the cursor changes from to . Otherwise a freehand rotation in any direction is possible.

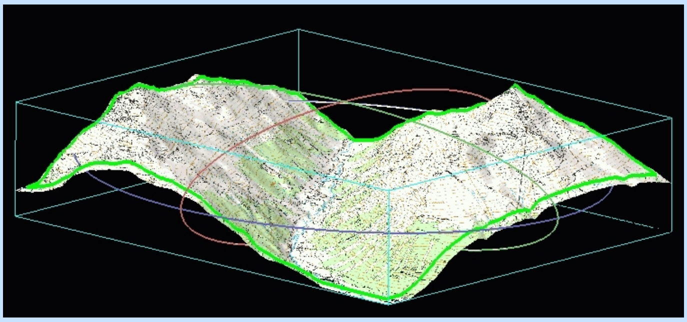


Figure 4.15: "Active" project with rotation axes.

End of exercise 4.4.b

Exercise 4.4.c: How to switch between 2D and 3D mode

- Click  to switch from 3D to 2D view. This button then changes to  and by clicking again, you will return to 3D view.

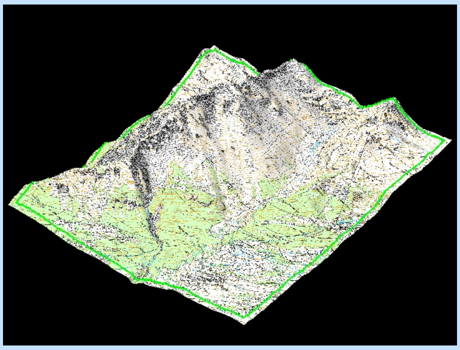


Figure 4.16: 3D view of example model.

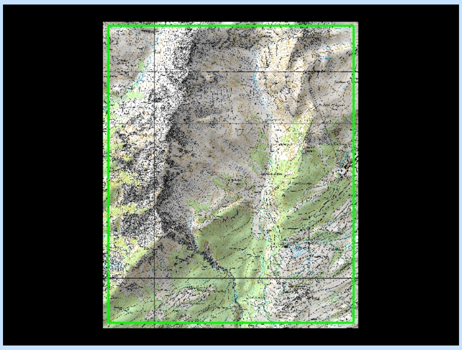


Figure 4.17: 2D view of example model.

In 2D-mode you have all possibilities that you know from 3D-mode. It works for input files as well as for simulations. For the following functions of RAMMS it is necessary to switch from 3D to 2D view:

INPUT:

- Draw New Release Area 
- Release area information 
- Crop release area 
- Draw New Forest Area 
- Draw New Domain 

OUTPUT:

- Draw New Line Profile 

End of exercise 4.4.c

4.4.2 Colorbar

As soon as a parameter is shown in the project, the colorbar appears on the right side of the main window. It can be turned on and off by clicking on .
The colorbar can be moved anywhere in the sreen (and can get lost). Use *Project→Get Colorbar* to find a lost colorbar.

Exercise 4.4.d: Editing the colorbar

Changing the minimum and maximum values of the colorbar as well as changing the number of colors used is done in the **Avalanche Panel** under **Display**.

- Simply type a new value into the respective field and hit the return key on the keyboard. The display will be refreshed.
- To view the underlying topography or image, you can change the transparency.
- **ATTENTION:**
  Values < x.xxx are not displayed!
  The cut off depends on the min and max values as well as on the number of colors. Make sure that you have the range of values you want to display!

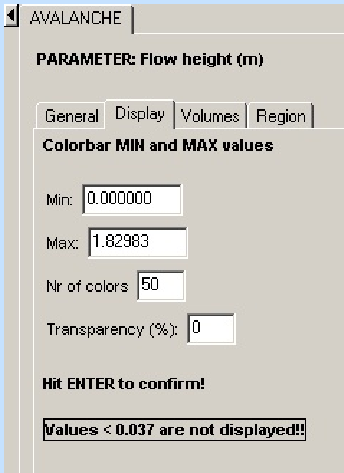


Figure 4.18: The display tab.

- Open the editing window by either choosing **Edit→colorbar properties** or clicking in the vertical toolbar.
- To change the colorbar properties simply click into the field you want to change, then click OK.

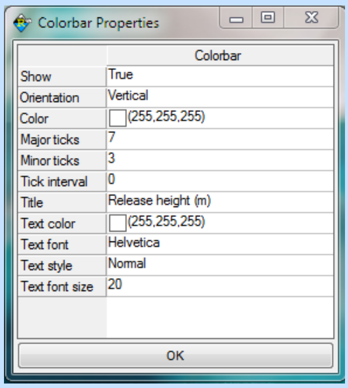


Figure 4.19: The colorbar properties window.

Under **Edit→Colorbar White Color** the text-color of the colorbar can be changed to white. This can be useful when changing the background color of your project to white **Track→Preferences→Avalanche tab→Background color** (see avalanche tab p.16).

End of exercise 4.4.d

4.4.3 Changing maps and remote sensing imagery

It is possible to change the map or imagery of a project anytime. Take into account, that the corresponding TFW-file (world-file) has to be in the same folder as the actual map (*.tif). If this is not the case, the map will not be found!

To check which map and imagery are currently loaded in the project, open the project input (or output) log (*Project→Input Log File*). Next to *map image* and *ortho image* you will find the location and name of the loaded map and imagery, respectively.

Exercise 4.4.e: How to add or change maps.

a) Add or change a map:

- Go to **Extras** → **Add/Change Map** or click .

  If more than one map is found, the following window pops up, listing the maps found:

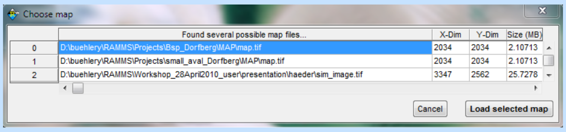


Figure 4.20: Window to choose map image.

  Information on the image dimensions (x-Dim and y-Dim, pixel) and size (in MB) are provided and might be a selection criterion.

- Select the map you wish to add and click **Load selected map**.

b) Map not found:

- If the question "No map found, continue search?" appears, you either don't have an appropriate map, the map-folder directory is set wrong or the map is saved in a different folder. In the latter case click **Yes** and choose the correct folder.
- Click **No** to cancel search or click **Yes** to continue search.
- A window pops up to browse for the correct map location and file.

End of exercise 4.4.e

Exercise 4.4.f: How to add or change remote sensing imagery.

- Go to **Extras**→**Add/Change Image** or click .
- ▶ See exercise *"How to add and change maps"* on page 32 above.

End of exercise 4.4.f

4.4.4 How to save input files and program settings

Once a project is created, it is saved under the name and location you entered during step 1 of the RAMMS::Avalanche Project Wizard (see figure 4.7 on page 25). The created input file has the ending *.av2.

The second situation in which the input file is saved automatically, is when a calculation is started. The saved input file has the same name as the created output file.

Exercise 4.4.g: How to save input files and program settings manually.

a) Input file:

- In case you want to save the input file manually before running a calculation, go on **Track** → **Save**. This is helpful, when a release area and MuXi-file was loaded but you wish to close the project before doing the calculation.
- If you wish to save a copy of your file under a new name, go on **Track** → **Save Copy As** or click .
- A window pops up to choose an old file which should be overwritten or to type in a new name, then click **Save**.
- Continue working on the original file, not the just saved one!

b) Program settings:

- If you have moved and/or rotated your project for a better view, you can save this position by going on **Extras** → **Save Active Position**.
- You can now get back to this position anytime by choosing **Extras** → **Reload Position**.

End of Example 4.4.g

4.4.5 How to open input and output files

Exercise 4.4.h: How to open an input file.

- Close any active project file.
- Go to **Track** → **Open...** → **Input File** or click .
- A window opens to browse for an avalanche input file (*.av2).
- Click **Open** after file name was selected.
- The project will be opened.

End of Example 4.4.h

Exercise 4.4.i: How to open an output file/avalanche simulation.

- Close any active project file.
- Go to **Track** → **Open...** → **Avalanche Simulation** or click .
- A window opens to browse for an avalanche simulation file (*.out.gz)
- Click **OK**.
- The simulation will be opened.

End of Example 4.4.i

Exercise 4.4.j: How to load an optional shapefile.

- To load a shapefile, click .
- A window opens to browse for a shapefile (*.shp).
- Click **Open** after file was selected.

End of Example 4.4.j

4.4.6 About RAMMS

Some information about the RAMMS installation on your computer is found here: *Help→About RAMMS...*.

Figure 4.21: About RAMMS...

4.5 Running a calculation

To run a calculation or a specific scenario within a newly created project (creating a project see section 4.3) it is necessary to define a release area, a calculation domain and friction parameters μ and ξ. While the definition of release areas and release heights as well as the set of friction parameters μ and ξ have a strong impact on the results of RAMMS simulations, the definition of a smaller calculation domain is especially usefull to keep the number of calculation points as small as possible. The exercises below show you how to create a release, a calculation domain and a MuXi-file. Details on the friction model used in RAMMS::Avalanche are given in section 4.2.3 on p.21.

4.5.1 Release area

There are different possibilities to include a release area into the project. The following table gives an overview of the possibilities RAMMS offers. For further explanations see the exercises below.

Create a new release area	If there is no release area available for your project, or you wish to create a new one, switch to *2D mode* and click .
Load an existing release area	Load an existing release area with *Input* $\rightarrow$ *Release area* $\rightarrow$ *Load existing release area*.
Import a shapefile and convert it to a release area	Draw a release area using a GIS-tool and save it as a polygon shapefile (.shp). Then convert the shapefile using *GIS* $\rightarrow$ *Convert Polygon Shapefile* $\rightarrow$ *Polygon Shapefile to RAMMS Release Shapefile*.

The definition of release areas and release heights have a very strong impact on the results of RAMMS simulations. Therefore we recommend to use reference information such as photography, GPS measurements or filed maps to draw release areas. This should be done by people with experience concerning the topographic and meteorological situation of the investigation area. Release areas can only be drawn in 2D mode.

Exercise 4.5.a: How to create a new release area.

- Switch to 2D mode by clicking .
- Activate the project by clicking on it once.
- Click .
- Click into the project where you want to start drawing the outline of the release polygon.
- Continue drawing the release polygon by moving the cursor and clicking the left mouse button.
- To end the release polygon, click with the **right mouse button**. The polygon will be closed automatically.

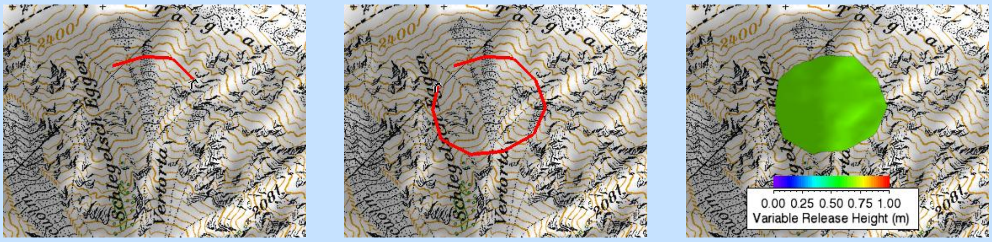


Figure 4.22: Project with emerging release area.

Before the release area is created, you have to answer a few questions:

- **Add more release areas?**
  You can either answer with **Yes** and create a second release polygon as explained above or answer with **No** and continue with the next step.
- **Choose a new release filename:**
  Enter a new name for the release area. The ending *rep.shp is added automatically.

The release area will now be created and opened directly, as well as the colorbar.

End of exercise 4.5.a

Exercise 4.5.b: How to load an existing release area.

- Choose **Input** → **Release area...** → **Load existing release area**.
- Select release file (*rep.shp) and click **open**.
  → The release area appears in the project as well as the colorbar for the variable release height (m)

End of exercise 4.5.b

Once a release area is created or loaded, you have to specify the release height. Switch to 2D mode, choose *Input* → *Release area...* → *Details/Edit release area* or click the button  and choose the release area polygon by selecting it with the left mouse button. The appearing window yields information about release area, mean slope angle, mean altitude and estimated release volume. And, most importantly, the release height can be entered, see exercise below.

Additional release information is found in the *Avalanche panel*, tab *Volumes*, see Fig. 4.23 below.

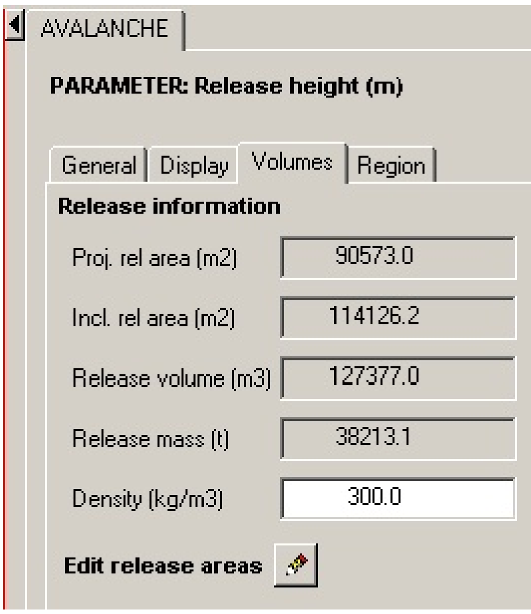


Figure 4.23: Release area and volume information.

Exercise 4.5.c: Specify release height and view release information

- Switch into 2D mode by clicking .
- Click on the **View/Edit release area** button (in the horizontal toolbar or in the volumes tab in the panel) or choose **Input** → **Release area...** → **Detail/Edit release areas**.
- Then click on the release area you want to get information on. A red polygon is drawn around the selected release area. The following window appears:

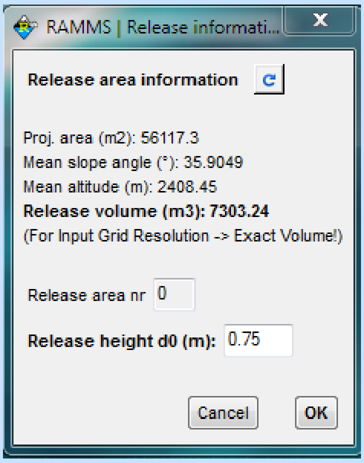


Figure 4.24: Release area information window.

Remark: The estimated release volume is very accurate for the grid resolution of your input project. If you calculate a different simulation resolution, the estimation can differ from the calculated release volume. If you do not trust the indicated values, click on the update button in the upper right corner (Release area information).

- To change the release height enter a new value (the resulting release volume is directly adjusted). Click **OK** if you want to keep the changes, **Cancel** otherwise.

End of exercise 4.5.c

4.5.2 Calculation domain

To reduce calculation time you can specify a smaller *calculation domain* to reduce the number of computational cells. By analyzing a calculation with a coarse grid (large cell size), e.g. with a cell size of 5 or 10 m, you get an idea where the flow path is situated and you can limit the calculation domain to the area of interest.

Switch to *2D mode* and choose *Input* → *Calculation Domain...* → *Draw New Domain* or click . Now you can draw a polygon containing the area of interest analogously to drawing a new release area (see exercise *"Create release area"* on page 36). We strongly recommend using smaller calculation domains especially if you calculate with small cell sizes (e.g. < 5m).

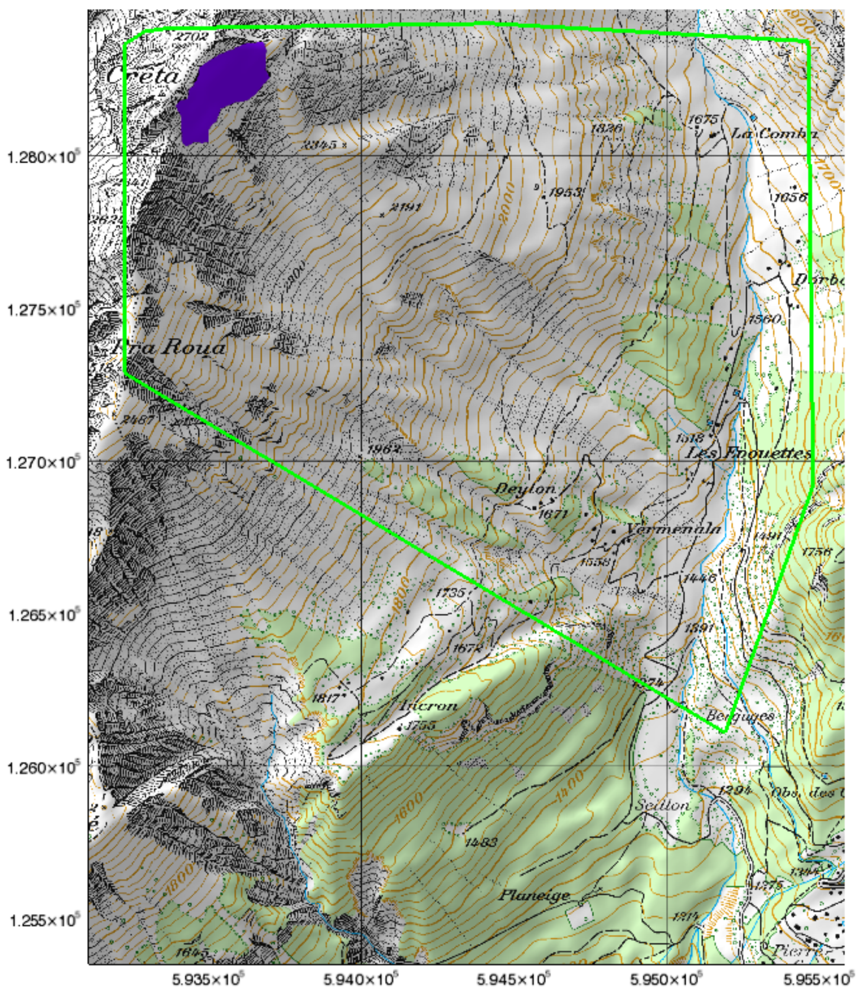


Figure 4.25: Calculation domain in green encloses the area of interest and reduces calculation time in comparison with the default rectangular domain which is automatically generated .

4.5.3 Friction parameters μ and ξ

Forest area

It is necessary to take forest areas into account, when running a simulation with variable friction parameters (μ and ξ).
The easiest way to consider forest, is to have a digital forest file (ASCII grid or forest shapefile) located in the folder set in the RAMMS preferences (FOREST directory). In this case the forest file will already be included while creating the project. Step 2 of the *RAMMS::Avalanche Project Wizard* deals with the DEM- and FOREST-files.

Exercise 4.5.d: How to create a FOREST file.

- Switch to 2D mode by clicking .
- Activate project by clicking on the map once.
- Click  or choose **Input** → **Forest...** → **Draw New Forest Area**.
- Trace the forest outline by creating as many FOREST area polygons as necessary (proceed as in exercise *"How to create a new release area"* on page 36) and name your new FOREST file. A new FOREST shapefile is saved.
- You are asked, if you want to import the created FOREST file into your project. Click **yes**, if you want to use the newly created FOREST (ignore the next point in this case). Otherwise click **no** and import the FOREST file later, as explained in the next point.
- Import the new FOREST shapefile: Choose **Input** → **Forest...** → **Import Forest Area From SHAPEFILE**, then select your FOREST shapefile.
- This new FOREST information is not automatically taken over in existing MuXi-files. Therefore, recreate existing MuXi-files if needed. If you create a new MuXi-file with **Input** → **Friction Values...** → **Create new MuXi File (Automatic Procedure)**, the forest will now be considered.

End of exercise 4.5.d

Global parameters

The friction parameters μ and ξ strongly depend on the volume and the return period of the avalanche. So prior to creating a new MuXi-file click *Input* $\rightarrow$ *Global parameters* and choose the return period and the volume category of the avalanche you would like to simulate. The MuXi-file will be calculated based on these values. Changing the return period and/or volume category has no effect on already existing MuXi-files. The default volume category is chosen based on the specified release volume.

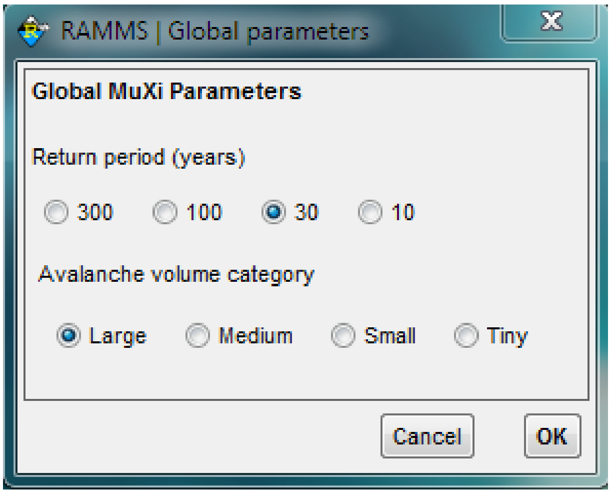


Figure 4.26: RAMMS global parameters.

MuXi-file

In RAMMS::Avalanche you can automatically generate μ and ξ for your calculation domain based on topographic data analysis, forest information and global parameters. The following exercises show you how to create and load MuXi-files for a RAMMS simulation with variable friction parameters.

Exercise 4.5.e: How to create a new MuXi-file.

- Choose **Input** $\rightarrow$ **Friction Values...** $\rightarrow$ **Create new MuXi File (Automatic Procedure)** or click .
- A window pops up where you have to define an appropriate return period and check your avalanche volume. You can also define these global parameters under (**Input** $\rightarrow$ **Global Parameters**).
- Enter a file name (e.g. Test).
- Unless you know better, leave the values as they are.
- Click **ok**.
- If this is the first MuXi-file for this project, or if you changed or removed a forest cover or if you changed the altitude limits when entering the file name, RAMMS will start a terrain classification. Otherwise, RAMMS will skip the terrain classification (the classification is saved in the file muxi_class.asc in the logfiles folder).
- The MuXi-file will be visualized after its creation. The μ- and ξ-values are saved in two asc-files (Test_mu.asc and Test_xi.asc respectively). Only the region within the calculation domain will be visualized.
- You can switch between the release area (if already loaded), and the μ and ξ values in the *choose Visualization* area in the avalanche panel.

End of exercise 4.5.e

Exercise 4.5.f: How to load an existing MuXi-file.

- Choose **Input** $\rightarrow$ **Friction values** $\rightarrow$ **Load existing MuXi-file**.
- A window opens to browse for an existing MuXi-file.
- Click **Open** and the file will be loaded.

End of exercise 4.5.f

4.5.4 Running a calculation

To run a calculation you have to open a created project (section 4.3), load a release area (section 4.5.1), and a calculation domain (section 4.5.2) must have been created and if a calculation with variable friction parameters is desired. A MuXi-file is necessary as well. Below you find two examples, one for running a constant calculation (constant release height and constant friction parameters μ and ξ) and one for using variable friction parameters.

Exercise 4.5.g: How to run a constant avalanche calculation.

- To run a calculation choose **Run** → **Run Avalanche Calculation** or click .
- The RAMMS::Run Simulation window opens. Before clicking **run calculation**, you should check the input parameters.

General

(1) Project name.

(2) Project info. You can change it by simply typing into the field.

(3) Additional information: calculation domain file and DEM file.

(4) Select an output filename.

(5) Check box **Run in background**: Option to run simulations in background mode. The RAMMS interface remains active and allows the user to start e.g. new simulations.

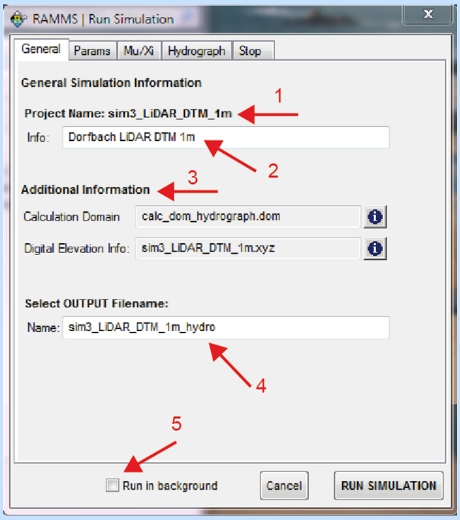


Figure 4.27: General information.

Continuation of exercise 4.5.g: How to run a constant avalanche calculation.

Parameters

Simulation Parameters:

(1) Change grid resolution, if necessary.
The resolution should always be chosen so that important features of the terrain are represented in the terrain model. High resolution grids will extend your calculation time.

(2) Choose end time of simulation.

(3) Choose dump-step interval.
The dump-step interval defines the resolution of the animation of your simulation but has no effect on the simulation results

(4) Keep the default value for density if no further informations on the avalanche density is available (300 kg/m^3).

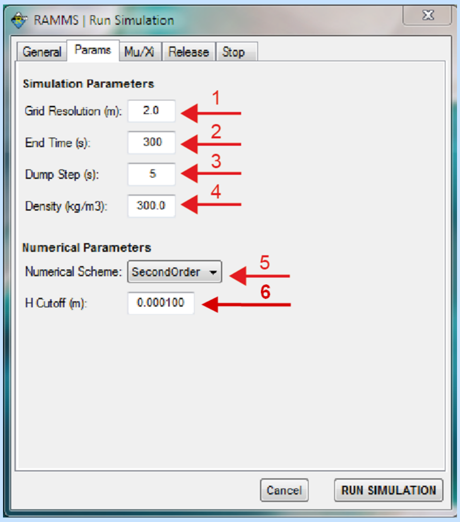


Figure 4.28: Calculation parameters.

Numerical Parameters:

(5) Change numerical solver, 1st or 2nd order scheme.
We recommend using 2nd order, because it provides more accurate solutions of the equations than 1st order. However if you encounter stability problems it may be useful to run your calculation using the 1st order numerical scheme.

(6) Keep the default value for the Null-height H cutoff (0.000100 m).
Unrealistic shallow flow heights of the simulation are eliminated to minimize numerical errors.

Continuation of exercise 4.5.g: How to run a constant avalanche calculation.

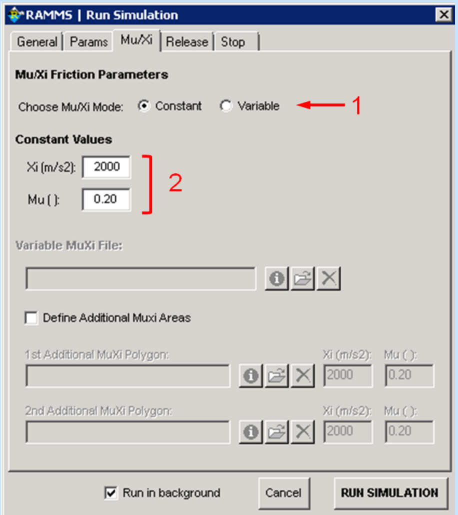


Figure 4.29: Friction values Mu and Xi.

Mu/Xi

(1) For a calculation with constant MuXi-values, click **constant**.

(2) Enter μ and ξ values. Choose **Help** → **RAMMS Manuals...** → **Friction Parameter Table (PDF)** or see friction value table in the appendix for an idea of μ and ξ.

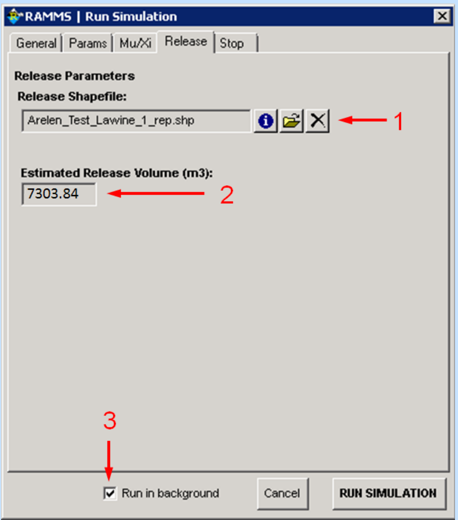


Figure 4.30: Release information.

Release

(1) The text field should indicate your release shapefile.

(2) The estimated release volume is stated in the second text field.

(3) Click the checkbox **Run in background** to run the simulation in the background.

Continuation of exercise 4.5.g: How to run a constant avalanche calculation.

Stop

The stopping criteria in RAMMS is based on the momentum. In classical mechanics, momentum p (SI unit kgm/s, or, equivalently, Ns) is the product of the mass and velocity of an object ($p = mv$). For every dump-step, we sum the momenta of all grid cells, and compare it with the maximum momentum sum. If this percentage is lower than a user-defined threshold value (see below), the program is interrupted and the flow is regarded as stopped. Threshold values between 1-10% are reasonable, but this is only a suggestion and has to be empirically determined for each test case.

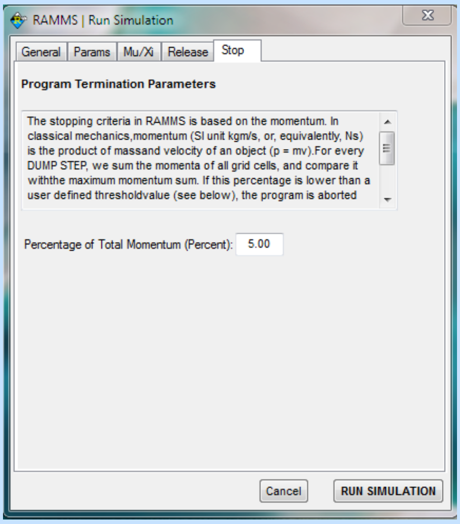


Figure 4.31: Stop criteria.

Continuation of exercise 4.5.g: How to run a constant avalanche calculation.

- Click **run simulation** (Fig. 4.30).
- If you want to start several (up to 50) simulations automatically (e.g. over night) use **Track→New...→Run Batch Simulations**. You can choose how many computational cores the Batch-Mode should use.
- The following window appears, showing the status of the calculation. (Fig. 4.32) (1) general information of the simulation, (2) output file, (3) starting the calculation (4) for every time step RAMMS calculates the maximal values (height, velocity and pressure) as well as the outflow mass, the moving momentum and the flow volume.

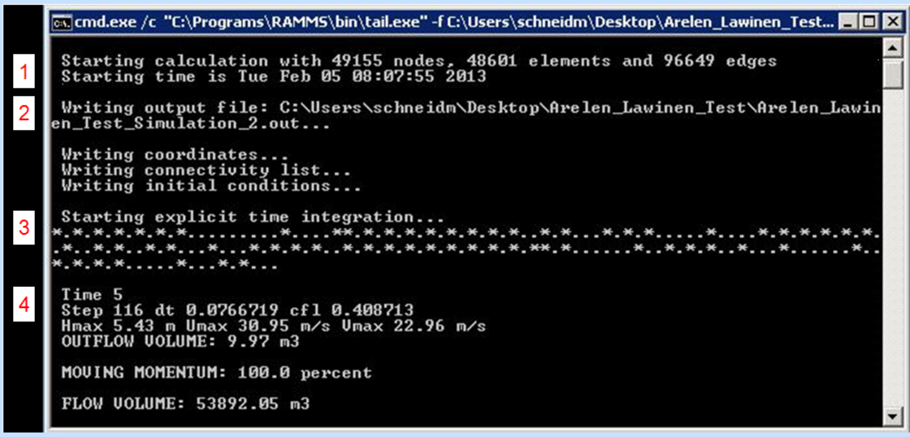


Figure 4.32: Status window of calculation.

Continuation of exercise 4.5.g: How to run a constant avalanche calculation.

- Once it's finished, the simulation as well as the output logfile (see Fig. 4.36) are opened in RAMMS. If you ran the simulation in backround mode, you have click on any button to finish the calculation. Afterwards the simulation is opened in RAMMS.

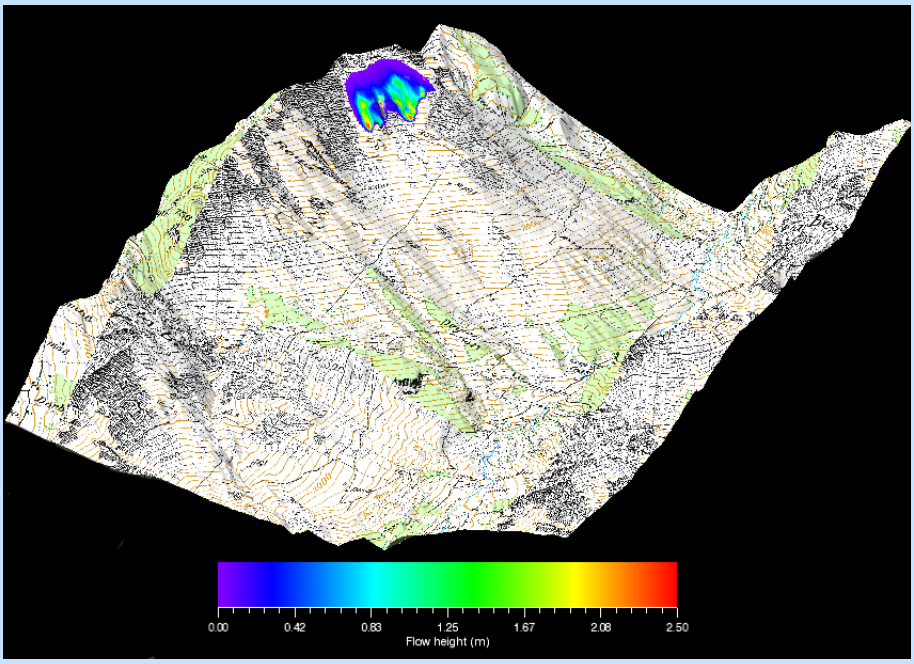


Figure 4.33: Main window in output mode.

- If mass flows out of the calculation domain, RAMMS shows an alert. (Fig.4.34). To get reliable results you should enlarge your calculation domain. (See section 4.5.2)

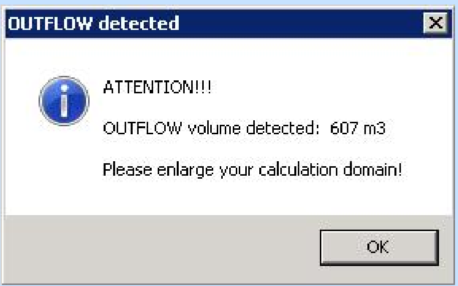


Figure 4.34: Outflow volume alert.

End of exercise 4.5.g

Exercise 4.5.h: How to run a variable avalanche calculation.

If you want to take the topography into account run a simulation with variable friction parameters μ and ξ. The process for a variable calculation is almost the same as for a constant calculation! Therefore, this exercise only shows the difference to the exercise before (exercise *"How to run a constant avalanche calculation"* on page 44).

Mu/Xi

(1) For a calculation with variable MuXi-values, click **variable**.

(2) Check, if the correct MuXi-file is used. If a different file should be used, click the open button  and browse for the desired file.

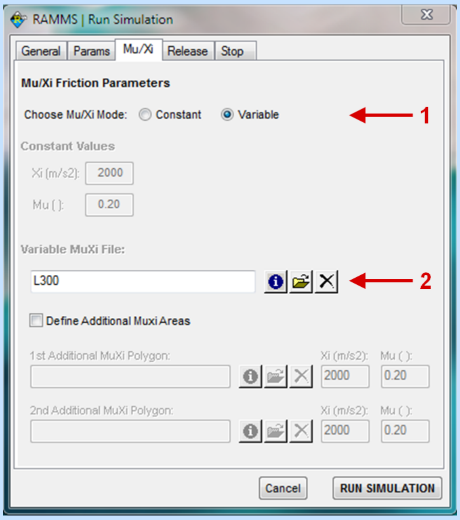


Figure 4.35: MuXi settings for a variable calculation.

End of exercise 4.5.h

4.5.5 Project information

Once a scenario within a specific project is calculated it is possible to open the *output logfile* (in output mode) including project settings and information as well as calculation specifications. You can open the project's output log with *Project→Output Log File*. A window as shown in Fig. 4.36 opens. This window provides information about your project and is the first thing to look at after running a simulation to check your simulation results.

(1) Information on simulation time and resolution. Be sure the *simulation stopped due to LOW FLUX*. Otherwise the output *TIME END CONDITION* informs you, that your simulation stopped before the avalanche reached the stopping criteria you defined for the simulation (see section Stop p.47).

(2) Information on simulation results.

(3) Input logfile (see Fig.4.37)

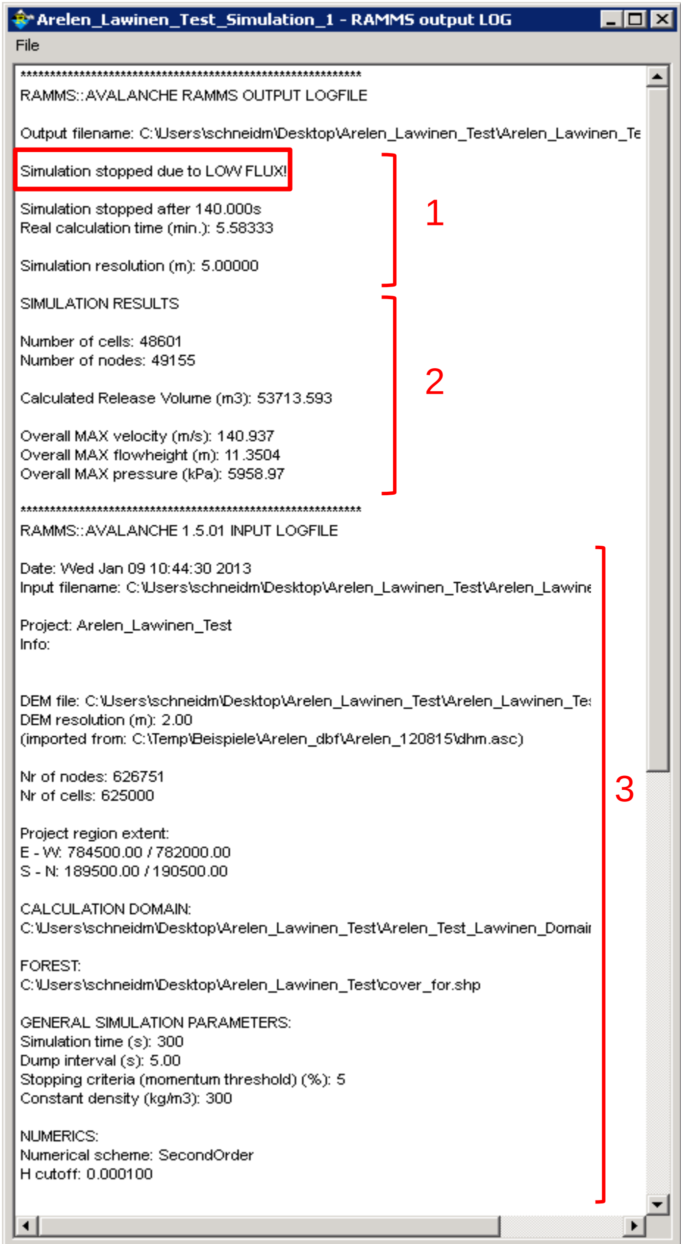

Arelen_Lawinen_Test_Simulation_1 - RAMMS output LOG
File
RAMMS::AVALANCHE RAMMS OUTPUT LOGFILE
Output filename: C:\Users\schneidm\Desktop\Arelen_Lawinen_Test\Arelen_Lawinen_Te
Simulation stopped due to LOW FLUX!
Simulation stopped after 140.000s
Real calculation time (min.): 5.58333
Simulation resolution (m): 5.00000
SIMULATION RESULTS
Number of cells: 48601
Number of nodes: 49155
Calculated Release Volume (m3): 53713.593
Overall MAX velocity (m/s): 140.937
Overall MAX flowheight (m): 11.3504
Overall MAX pressure (kPa): 5958.97
RAMMS::AVALANCHE 1.5.01 INPUT LOGFILE
Date: Wed Jan 09 10:44:30 2013
Input filename: C:\Users\schneidm\Desktop\Arelen_Lawinen_Test\Arelen_Lawine
Project: Arelen_Lawinen_Test
Info:
DEM file: C:\Users\schneidm\Desktop\Arelen_Lawinen_Test\Arelen_Lawinen_Te
DEM resolution (m): 2.00
(imported from: C:\Temp\Beispiele\Arelen_dbf\Arelen_120815\dhm.asc)
Nr of nodes: 626751
Nr of cells: 625000
Project region extent:
E - W: 784500.00 / 782000.00
S - N: 189500.00 / 190500.00
CALCULATION DOMAIN:
C:\Users\schneidm\Desktop\Arelen_Lawinen_Test\Arelen_Test_Lawinen_Domai
FOREST:
C:\Users\schneidm\Desktop\Arelen_Lawinen_Test\cover_for.shp
GENERAL SIMULATION PARAMETERS:
Simulation time (s): 300
Dump interval (s): 5.00
Stopping criteria (momentum threshold) (%): 5
Constant density (kg/m3): 300
NUMERICS:
Numerical scheme: SecondOrder
H cutoff: 0.000100

Figure 4.36: Output Logfile.

The input logfile (included in the output logfile), however, can already be opened once a project is created and before a simulation is performed.

There are two ways to view your project settings and information. First you can open your project's input logfile (or output logfile, in *output* mode), or you can check your project's region extent and area in the avalanche panel in the region tab.

You can open the project's input log file with *Project→Input Log File*. The following window opens:

This window provides information about all your project's input specifications, like number of nodes and cells, release areas, forest files, which DEM was used, the loaded map and ortho images as well as your global simulation parameters.

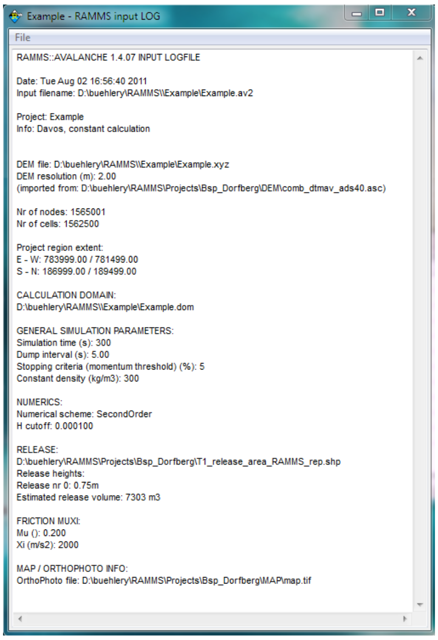


Figure 4.37: RAMMS Project Input Log file.

To view the project coordinates, click the region tab in your avalanche panel. The region tab lists X- and Y-Coordinates of the lower left (minimal values) and upper right (maximal values) corner (this are the coordinates you entered when creating the project) as well as the global minimum and maximum of altitude (Z value). Additionally, the total region area is shown (in km^2).

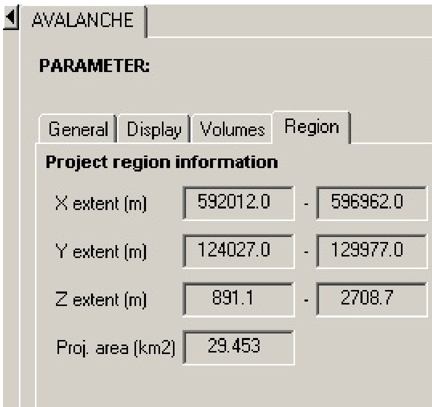


Figure 4.38: Region extent (X-, Y- and Z-Coordinates, total area).

4.6 Visualization and analysis of the results

This section gives a short overview on what is possible in RAMMS to view and analyze the simulation results. The interpretation of the results has to be done by an expert who is familiar with the local as well as with the topographic and meteorological situation of the investigation area.

RAMMS is a model and each model is a simplification of reality, therefore the simulation results should not be analized without questioning them. We strongly recommend that all users perform sensitivity studies.

4.6.1 Visualize different parameters

The drop down menu *Results* offers the following functions:

- Flow Height
- Flow Velocity
- Flow Pressure
- Flow Momentum
- Max values (Height, Velocity, Pressure, Momentum)
- DEM Adaptations (Add Deposition to DEM)
- Flow Analysis (Summary of Moving Mass)
- Friction Values (μ, ξ)

These results are all visualized by a colorplot in the topography. See exercise *"Displaying max values"* on page 55.

4.6.2 Line profile and time plot

In the horizontal toolbar you find two further functions:

- Line Profile
- Time plot

Line profile

A line profile is a good alternative to the color plot, if the avalanche snow height, velocity or pressure should be known at a specific location. The graph shows the currently active parameter. Every line profile is saved in the file *profile.txt* in the project directory. If you want to keep this line profile, you have to save it, see exercise *"How to draw a line profile"* on page 56.

Time plot

This function provides a time plot at a single point. This is helpful when it is of interest to know the values and maximum values at a specific location (e.g. at a building, dam, or a tree) through time. Every point is saved in the file *point.txt* and a point-info file *point_info.txt* is additionally saved in the project directory. If you want to keep this point, you have to save it, see exercise *"How to create a timeplot"* on page 58. The point-info file can be visualized with *Extras*→*Point...*→*View Point Info File*.

Exercise 4.6.a: Displaying calculation values.

The maximum values of snow height, velocity and pressure give a good overview of the dimension of the avalanche. You find them under
Results → **Max values...**

→ **Max flow height**
→ **Max velocity**
→ **Max pressure**

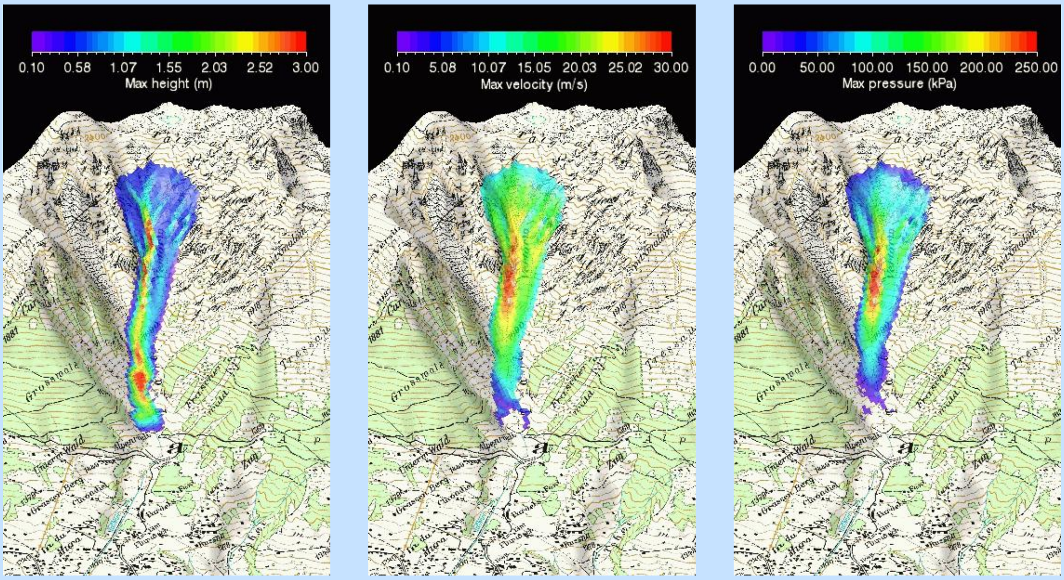


Figure 4.39: Results: Maximum values of flow height (left), velocity (middle) and pressure (right).

The flow height can be visualized exaggerated by a factor. Click **Help** → **Advanced...** → **Additional Preferences...** → **Edit** to change the factor of the quasi 3D-visualization of the flow height under the keyword **exaggeration**.

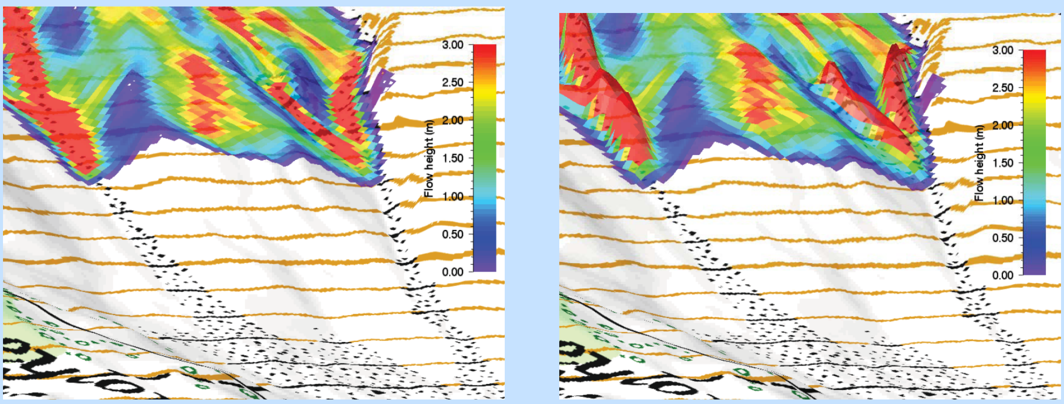


Figure 4.40: Quasi 3D-Visualization of flow height (left: exaggeration 1; right: exaggeration 5)

End of exercise 4.6.a

Exercise 4.6.b: How to draw a line profile.

a) Draw a new line profile:

- Switch to 2D mode by clicking .
- Activate the project by clicking on it once, then click or choose **Extras** → **Profile...** → **Draw New Line Profile**.
- Define the line profile in the same way you specify a new release area. Finish the line profile with a right-click on the mouse button.
- A window opens, displaying the line profile.

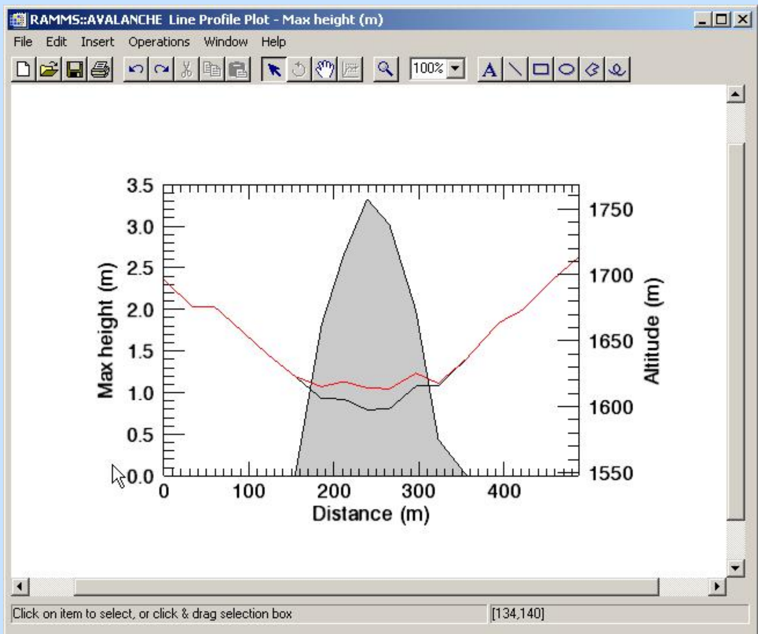


Figure 4.41: Line profile plot.

- filled grey area	active parameter (scale on left side).
- red line	active parameter (multiplied by 50) added to the track profile (altitude, scale on the right side).
- black line	track profile (altitude, scale on the right side).
- bottom scale	projected profile distance (in m).

- If you change the active parameter, min or max values or the dump-step in RAMMS, the plot is directly updated. You can also start the simulation and then watch the time variations in your line profile plot.
- It makes sense to either draw a profile line perpendicular to the flow direction or to draw the line along the avalanche path. Basically every imaginable path is possible.

Continuation of exercise 4.6.b: How to draw a line profile.

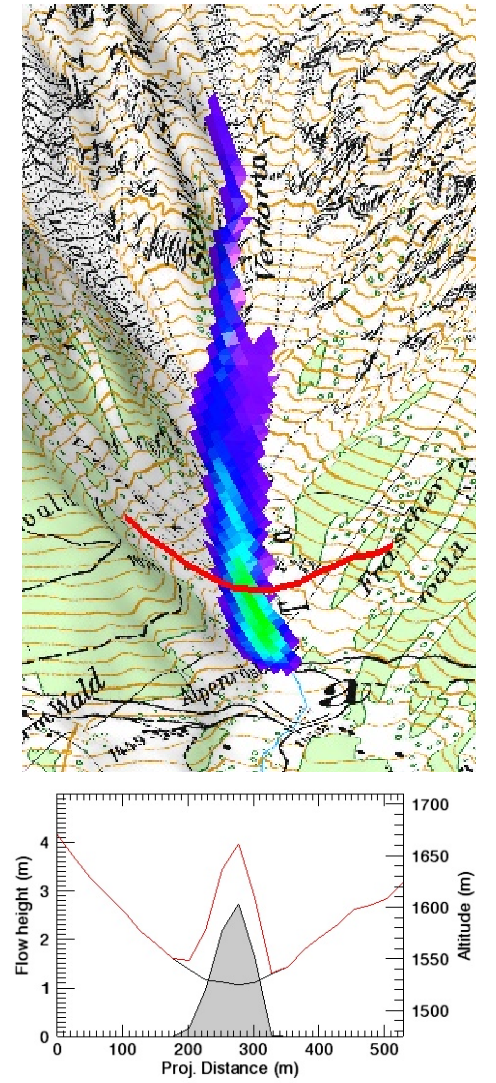


Figure 4.42: Line profile perpendicular to the flow direction.

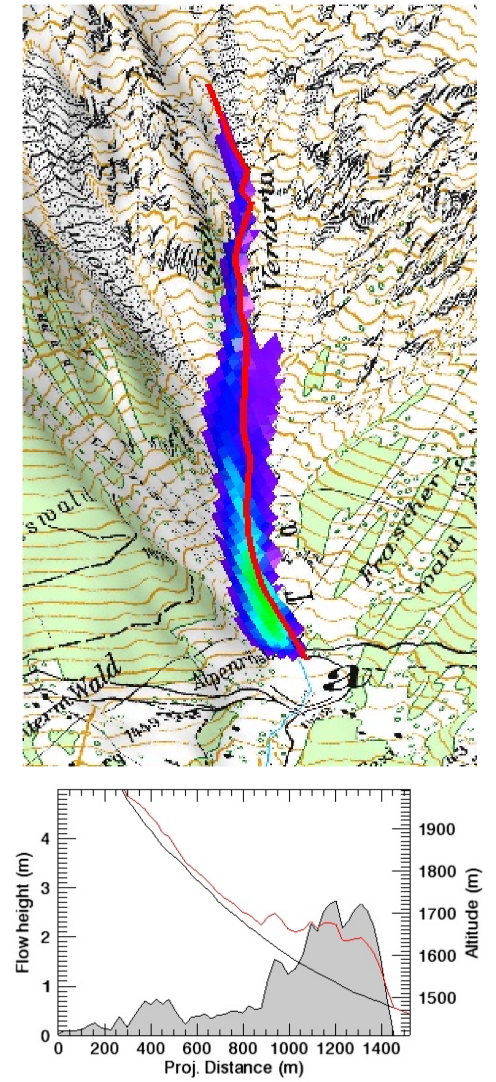


Figure 4.43: Line profile along the avalanche path.

- To save the coordinates of the points belonging to the line profile, go on **Extras** → **Profile...** → **Save Line Profile Points** and enter a file name.
- To save the line profile parameters (distance in m and the active parameter, e.g. the flow height in m) at the current dump-step, go on **Extras** → **Profile...** → **Export Profile Plot Data** and enter a file name.

Continuation of exercise 4.6.b: How to draw a line profile.

b) Load an existing line profile:

- Switch to 2D mode by clickin .
- Activate the project by clicking on it once and click or choose **Extras** → **Profile...** → **Draw New Line Profile**.
- Click the **middle mouse button** once.
- A window pops up and you can browse for the line profile you wish to open.

End of exercise 4.6.b

Exercise 4.6.c: How to create a time plot.

a) Select time plot point:

- Click or choose **Extras** → **Point...** → **Choose Point**.
- Click into the map at the point where you want to create a time plot.
- A window opens, displaying the time plot at the point of interest (active parameter vs. time).

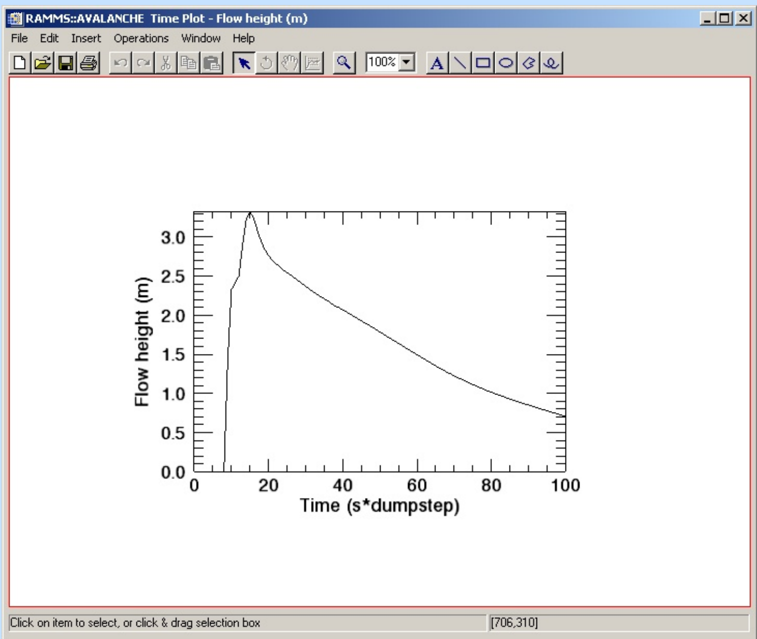


Figure 4.44: Time plot window.

Continuation of exercise 4.6.c: How to create a time plot.

- To save the point coordinates, choose **Extras** → **Point...** → **Save Point Location** and enter a file name.
- To save the time plot data (time in s and the active paramter, e.g. the flow height, for every dump-step), choose **Extras** → **Point...** → **Export Point Plot Data** and enter a file name.

b) Load a time plot:

- To reopen the time plot graph window of the last selected point, go on **Extras** → **Point...** → **Create Point Time Plot**.

- To open an arbitrary time plot that was saved anytime before, click .
- Click the middle mouse button once.
- A window pops up and you can browse for the time plot file you wish to open.

End of exercise 4.6.c

Exercise 4.6.d: Enter point coordinates and get a time plot.

- Go to **Extras** → **Point...** → **Enter Point Coordinates (X/Y)**.
- Enter X-coordinate of your point of interest. Click **OK**.
- Enter Y-coordinate of your point of interest. Click **OK**.
- The time plot opens.

End of exercise 4.6.d

4.6.3 Creating an image or a GIF animation

Image

It is possible to export your results as an image in different formats (e.g. .png, .jpg, .gif, .tif etc.). Click or choose *Track* → *Export...* → *Image File* and define a new file name with the corresponding extension. An image of the visible part in the viewer will then be saved.

GIF animation

Creating a GIF animation is only possible in output mode.
Click or choose **Track** → **Export...** → **GIF Animation** and wait until the simulation stopped and a window opened. Enter a file name and location. The GIF animation folder as well as the corresponding gif animation file are saved in the simulation folder. In the *avalanche* tab in the *preferences* you can define the interval for the GIF animation (GIF animation interval [s]). Only time steps, not maximum values can be saved as GIF animations.

4.6.4 Stopping mechanism

Check the output logfile under *Project* → *Output Logfile* to verify your simulation stopped due to low flux (see Output Logfile on page 51). Otherwise enlarge the end time of your simulation (see exercice *"Run a calculation"* on page 45.) To check the stopping of your simulation click *Results* → *Summary of Moving Mass*. A window similiar to Fig. 4.45 opens which shows the summary of moving mass. For every dump-step, RAMMS summed up the momenta of all grid cells, and compared it with the maximum momentum sum. If this percentage is smaller than a user defined threshold value (see page 47), RAMMS abortes the simulation and the avlanche is regarded as stopped.

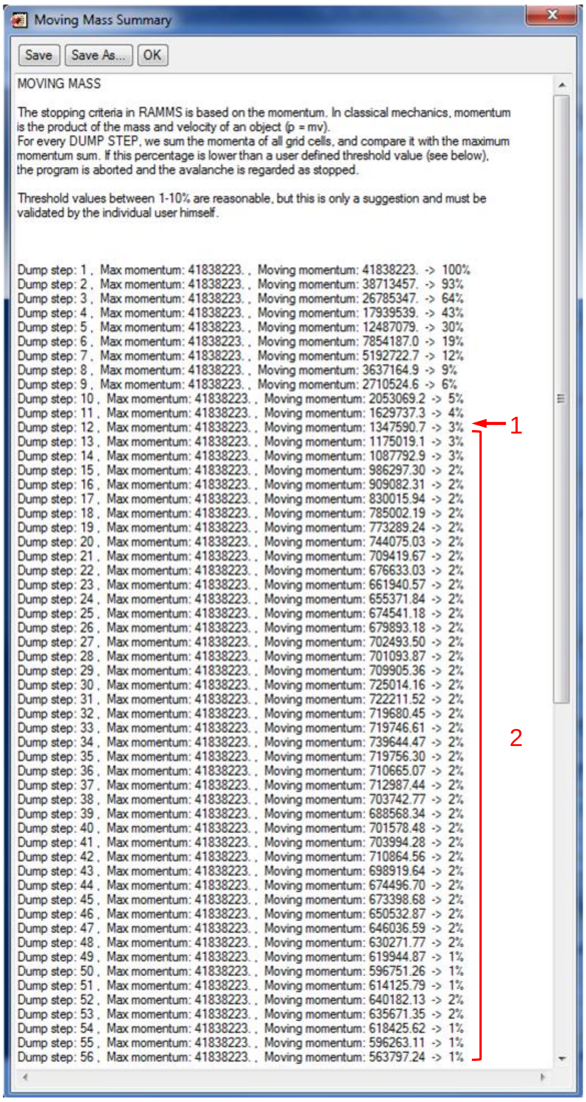


Figure 4.45: Summary of moving mass.

Stopping criteria with large treshold values (e.g. >10%) may result in unrealistically early stopping of a simulation.

Small threshold values however may lead to numerical diffusion of the simulation results as shown in Fig. 4.45 (2) and very slow creeping of the avalanche and velocity oscillations (see Fig. 4.46).

In the example shown in Fig. 4.45 the simulation could be already stopped after 12 dump-steps (1).

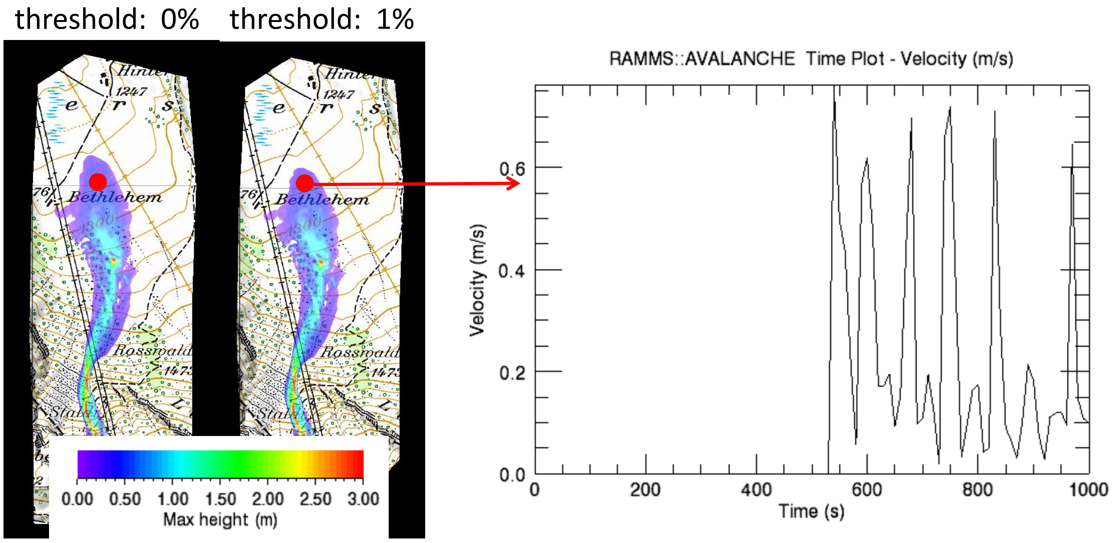


Figure 4.46: Stopping behavior of a RAMMS simulation. Small threshold values may lead to unlikely slow creeping of the material. In the examples shown in the figure above the stopping criteria is set to 0% respectively 1%.

Whether or not an avalanche stops depends on terrain (slope angle in runout), total flow volume and friction values and should always be evaluated by an expert. In case of doubt on how to choose threshold values we recommend running a simulation with a 1% threshold and checking the summary of moving mass for numerical diffusion (Fig. 4.45) and analysing the avalanche runout (flow height and flow velocity) with time plots (Fig. 4.45 and section 4.6.2).

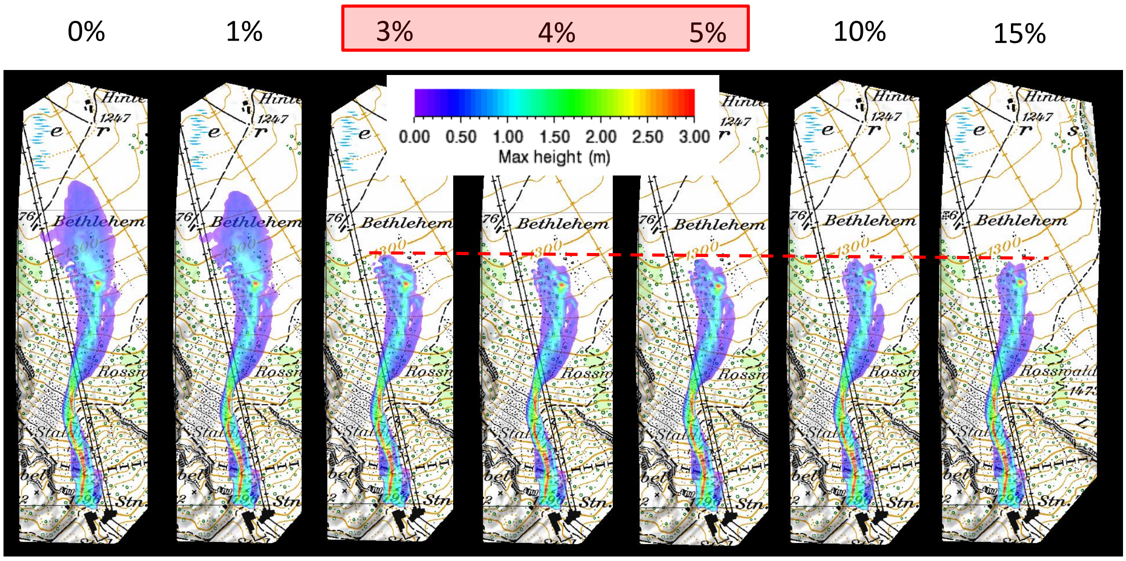


Figure 4.47: Stopping behavior of a RAMMS simulation. In this example threshold values <2% lead to numerical diffusion of the simulation results. Threshold values between 3-5% seem to be appropriate.

4.6.5 Numerical instabilities

Numerical instabilities can occur in RAMMS simulations (see Fig. 4.48).
To detect them plot the maximal flow velocity (click *Results* → *Max Values...* → *Max Velocity* or V)

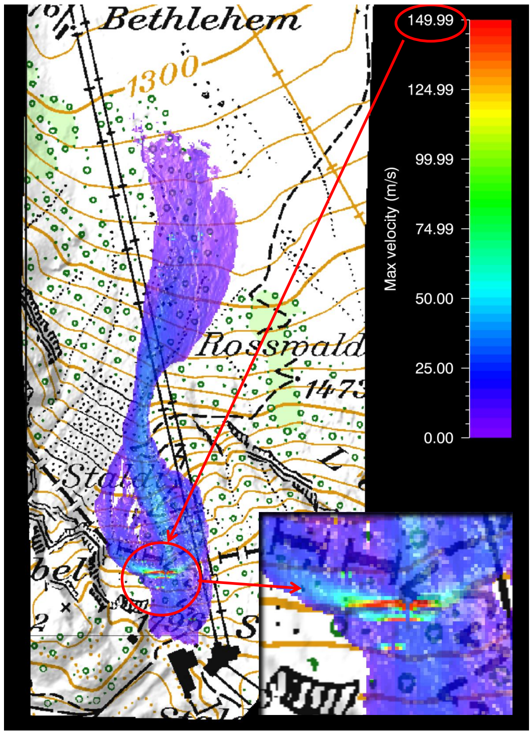
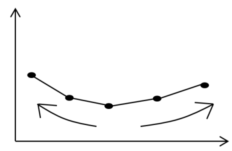
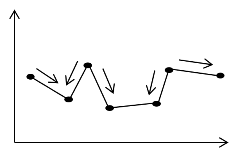


Figure 4.48: Numerical instabilities.

Numerical instabilities can happen because RAMMS employs 2nd-order numerical solution methods which can lead to problems, especially when topographic changes are large.



smooth terrain: there is no problem. Spatial gradients are constructed over several neighbour elements.



rough terrain: numerical instabilities may arise because it is difficult to construct reasonable gradients, especially velocity gradients over several neighbour elements. This can result in unrealistic velocity peaks in the solution.

There are different solutions to numerical instabilities (see Fig. 4.50):

If velocity peak	**Action**
• is local	→ ignore the outliers
• has propagated away from the source	→ smooth the terrain
	(mean filtering e.g. in ArcGIS Fig. 4.49) or
	→ run a 1st order calculation in RAMMS

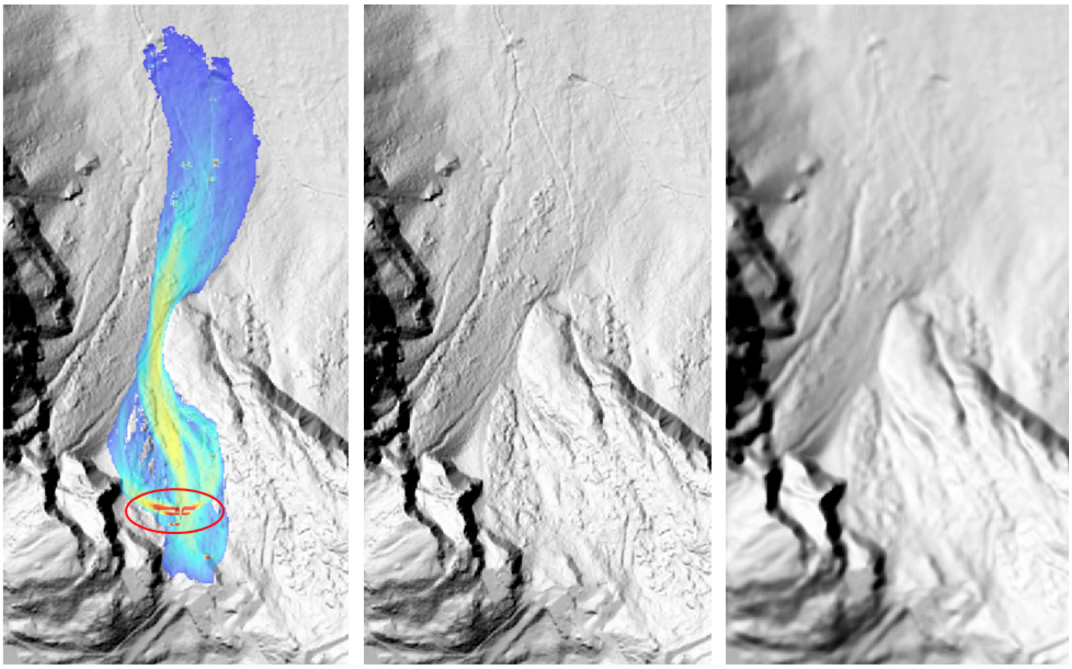


Figure 4.49: Smoothed terrain (e.g. mean 5x5 filter in ArcGIS).

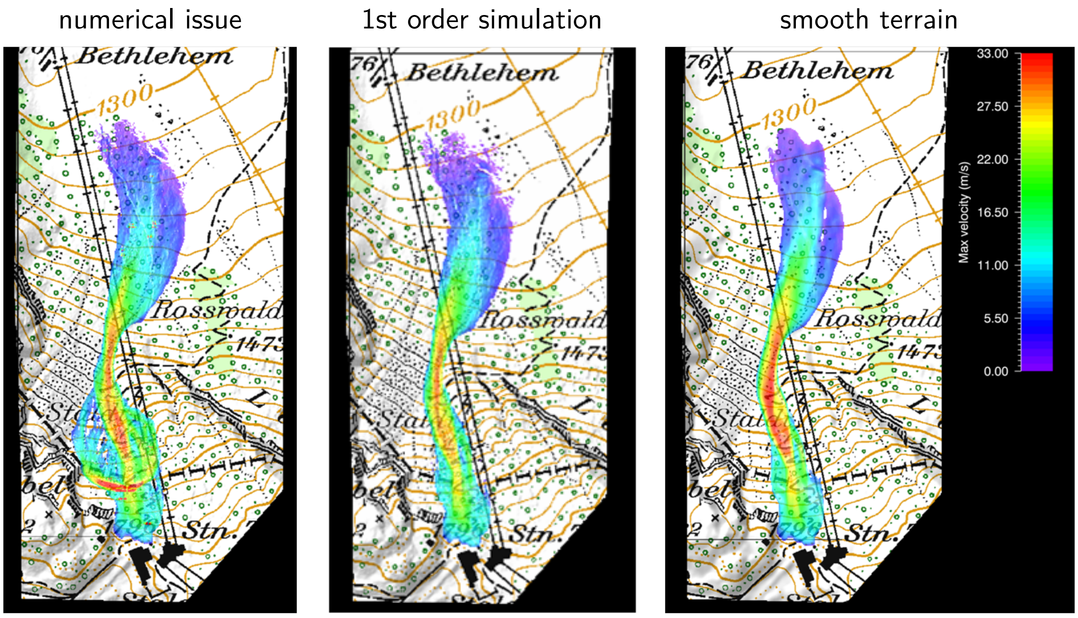


Figure 4.50: Solutions to numerical instability problems.

4.7 Adding structures or deposition to DEM

The option of adding structures or deposition to DEM must be used with great care and should not be used to design deflecting dams. Deflecting or catching dams can neither be designed directly with RAMMS nor can the residual risk below dams be calculated directly with RAMMS. RAMMS takes important factors in dam design such as energy dissipation, dam geometry or snow deposits in front of a dam not properly into account. Dams have to be designed using well known standard engineering procedures, e.g. Johannesson et al. 2009 [1] and Rudolf-Miklau and Sauermoser 2011 [2]. RAMMS is well suited to calculate the key input factors for dam design such as flow height and velocity. The dam-option should however only be used to try to visualize the influence of guiding or small deflection of the avalanche mass. RAMMS cannot be used directly to evaluate if the height of a deflecting dam is sufficient for a certain scenario or not (see explanations on page 67).

4.7.1 Creating a dam

RAMMS offers the possibility to simulate the presence of a deflecting dam by increasing the altitude at the position where a dam is considered. This option helps the user to design mitigation structures and to test its influence on potential flow paths near populated areas.

Exercise 4.7.a: How to create a new DEM to simulate a dam.

- Create a polygon ("release area") where a dam is supposed to be built (Fig. 4.51).
- Create a second, inner polygon, if you wish to have a two-stage dam.
- Go on **GIS→Add DAM to DEM...**.
  You have two options **...→Enter Relative Dam Height** or **...→Enter Dam Elevation**
- You will be asked to "Open dam file (*.rel)". Select the shapefile you want to use as the outer edge of the dam.
- The question pops up, if you want to "Open 2nd dam shapefile (inner polygon)?"
  - Click **No** to continue with the next step.
  - Click **Yes** to choose 2nd dam file (*.rel).

Continuation of exercise 4.7.a: How to create a new DEM to simulate a dam.

- Next step is to enter the total elevation height or the total relative height of the dam in meters. This is the elevation of the dam crest.
- If you loaded an outer polygon file, you will be asked to enter the intermediate height (m) (height of the outer polygon file) as well.
- Finally you have to "Enter new DEM name". Your new DEM, containing the "dam" is created in the folder set as DEM directory (RAMMS preferences ).

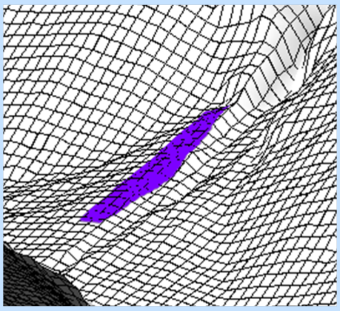


Figure 4.51: Release area where a dam is supposed to be built.

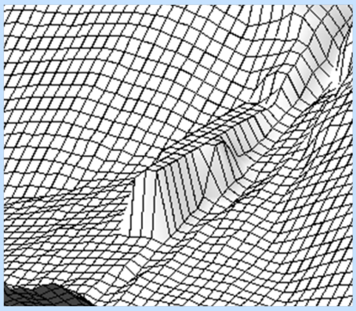


Figure 4.52: Dam.

End of exercise 4.7.a

To run a simulation based on the new created DEM, you first have to create a new project. Do almost exactly the same as if creating a regular project without the dam information. The only important difference is that you have to choose the correct DEM-file manually during step 2 of the project wizard.

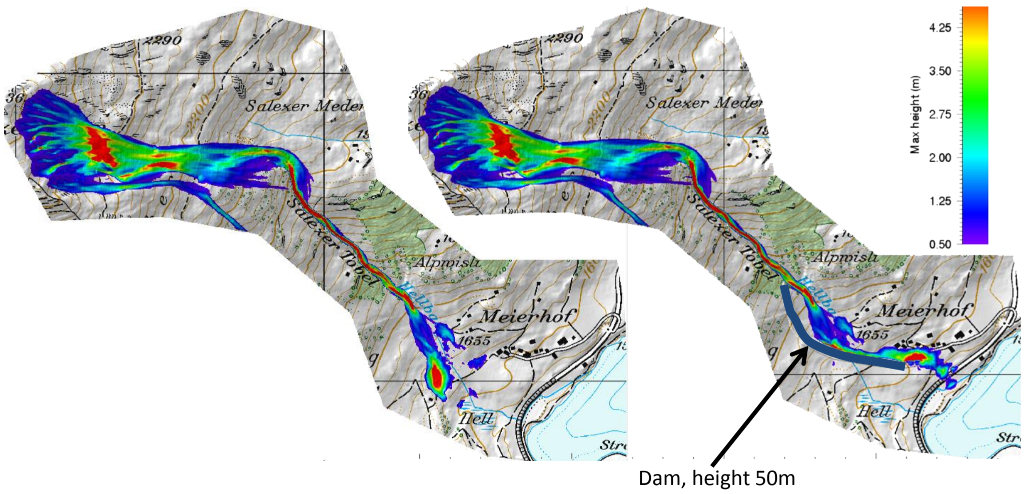


Figure 4.53: Simulation without (left) and with (right) a dam.

While RAMMS is able to simulate the effect of a dam lying lateral to the direction of flow quite well, there might occur numerical problems if a dam lies perpendicular to the direction of flow.

• Because there is no energy dissipation due to collision with dams implemented in RAMMS, unrealsitically large flow velocities and flow heights may be simulated in front of a dam.

• The numerical solver used in RAMMS incorporates information from neighboring cells. The effect of dams with only one cell as dam side wall may therefore be difficult to simulate.

If you encounter problems with the simulation of mitigation measures as described, we suggest creating a DEM including a dam in GIS ideally using progressively increasing side walls as shown in Fig. 4.54.

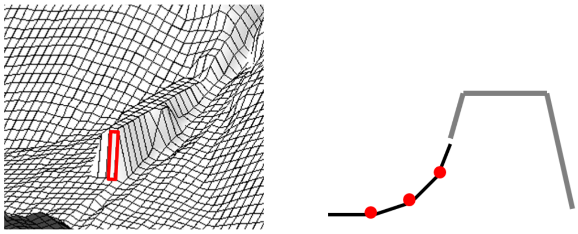


Figure 4.54: Dam with gradually rising side walls.

The interpretation of RAMMS simulations including mitigation measures such as dams has to be done by experts. In addition we recommend to always check the simulation results with engineering approaches.

4.7.2 Creating a new DEM with avalanche deposition

In case you wish to simulate an avalanche overflowing a previous avalanche, you should take into acount the deposition of the previous avalanche, because the path of the second avalanche will be influenced by the modified terrain. In RAMMS one has to assume that the deposits from an initial avalanche are not entrained by a subsequent avalanche. To do this, in the output mode, users can select the option to add the flow height of an avalanche to the DEM at any arbitrary dump-step. Finally, a new project can be created based on the updated DEM.

Exercise 4.7.b: How to add avalanche deposition to new DEM.

- The deposition height is the flow depth at the end of a simulation when the avalanche is considered to have stopped moving (alternatively, earlier dump-steps may be used if there are reasons to believe the flow should have stopped earlier). So first view the results at the last time step or a different time step, if desired.
- Go to **Results**→**DEM Adaptations**→**Add Deposition to DEM**.
- Enter a new name for the new DEM.
- The new DEM, containing the deposition information, is created. To run a simulation based on this DEM create a new project and manually choose the DEM file during step 2 of the wizard as explained above for the dam.

End of exercise 4.7.b

5 Program Overview

RAMMS is a windows-based program that relies on drop-down menus and dialog boxes to set the model parameters, run calculations and view results. Toolbar buttons are also available and provide short-cuts of the menu paths; moving the cursor over a button results in a short explanation, appearing in a text box below the cursor ('tooltip'). For functions not available in the current context, the menus and buttons are deactivated and cannot be used.

5.1 The Graphical User Interface (GUI)

The graphical user interface (GUI), see figure 5.1 below, consists of menu bar, horizontal and vertical toolbar, main window, time step slider, right and left status bar, colorbar and panel. They will be explained in the following sections.

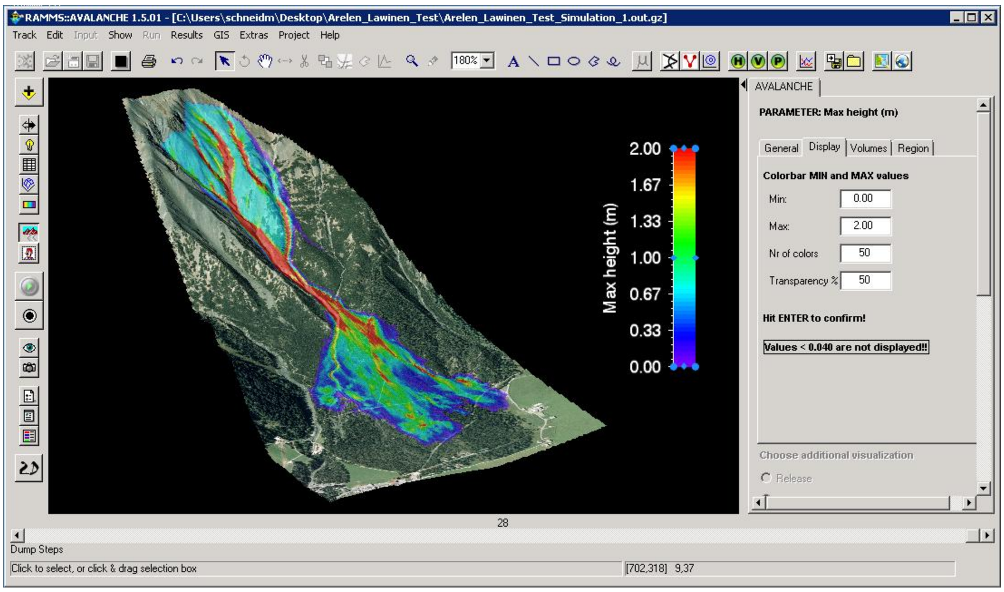


Figure 5.1: Graphical user interface (GUI).

5.1.1 The menu bar

Track

Similar to the Microsoft Windows *File* menu, *Track* is used to open, close, save, print, backup and export files.

New...	▶ **Project Wizard**	Start a new project, guided by the wizard (Ctrl+w).
	▶ **Convert XYZ → ASCII grid**	Convert laser scanning data into a ESRI ASCII grid.
	▶ **Run BATCH simulations**	Possibility to start up to 50 simulations automatically (e.g. overnight). You can choose how many computational cores the Batch-Mode should use (quasi parallel simulations, saves computionals time).
Open...	▶ **Input File**	Open an existing input file (*.hl2) (Ctrl+O).
	▶ **Avalanche Simulation**	Open existing avalanche simulation (select the folder containing the simulation files) (Ctrl+A).
Close		Close active file (input or output).
Save		Save active file (Ctrl+S).
Save Copy As		Save a copy of the active file (e.g. *test.hl2*) under a new name (e.g. *simulation1.hl2*, works only in input mode). But, RAMMS stays with the active file (*test.hl2*)!
Export...	▶ **Export Image File**	Create an image of the active window in a chosen formate. You can choose the desired image format using the file extension (e.g. .png, .jpg, .gif, .tif etc.).
	▶ **Export GIF Animation**	Create a GIF animation of the simulation (only in output mode). Change GIF animation interval (s) in the preferences.
Backup...	▶ **Backup RAMMS Version**	Make a backup of the current RAMMS version.
	▶ **Backup Active Project**	Backup your active project. The user will be asked if he wants to include output files in the backup. This function is useful when having problems with a simulation. Make a backup and send the zip-file together with some explanations to ramms@slf.ch. Make sure that all your input data (release area shapefiles, domain files, etc...) is in the project folder.
	▶ **Backup User Defined Files/Folders**	Backup any folder or files you want.
Preferences		Change RAMMS preferences.
Log files...	▶ **RAMMS Logfile (current)**	Show active RAMMS logfile.
	▶ **RAMMS Logfile (last session)**	If RAMMS crashed, open this logfile and copy/paste the content into an email to ramms@slf.ch.
Exit		Exit RAMMS (Ctrl+Q).

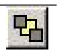
Edit

This menu is used to edit colorbar and dataspace properties. It is active only in input mode.

Edit Colorbar Properties	Edit the colorbar properties.	
Edit Dataspace Properties	Edit your dataspace properties.	
Show Dataspace Axes	Shows or hides dataspace axes of the project region. The axes can be only visible if the background color is set to white.	
Colorbar White Color	Defines the text-color of the colorbar (black or white).	

Input

Menu used to specify the global parameters, the calculation domain, release area, friction parameters and forest cover. This menu is active only in input mode.

Calculation Domain...	▶ **Draw New Calculation Domain**	This activates the button to draw a new calculation domain. The mouse cursor changes to an arrow.	
	▶ **Load Existing Domain**	Load a existing calculation domain (*.dom) drawn and saved before.	
Release Area...	▶ **Draw New Release Areas**	This activates the button to draw new release areas. The mouse cursor changes to an arrow.	
	▶ **Load Existing Release Areas**	Load an existing release area shapefile.	
	▶ **Details/Edit Release Areas**	The mouse cursor changes to an arrow and you can select release areas to define the release height and to view release area information. This works only in **2D mode**.	
	▶ **Crop Release Area**	If your release area shapefile consists of several polygons, you can crop some of them and create a new release shapefile.	
Friction Values...	▶ **Load Existing MuXi File**	Load afore created MuXi file (*muxi.shp).	
	▶ **Create New MuXi File (Automatic Procedure)**	The DEM is analysed, classified and according to altitude, slope and curvature information, a new MuXi-file is created.	
	▶ **Show MuXi Classification**	Shows the result of the MuXi-classification.	
Forest...	▶ **Draw New Forest Area**	The mouse cursor changes to an arrow and you can draw new forest areas. Save the forest shapefile and then use the function below (Import Forest Area From SHAPEFILE) to import the forest cover.	
	▶ **Show Active Forest Cover**	If forest cover is taken into account, the corresponding shapefile is displayed. If your project uses no forest cover at the moment, RAMMS will tell you so.	
	▶ **Import Forest Area From SHAPEFILE**	If a forest shapefile has been drawn it can be imported using this function.	
	▶ **Import Forest Area From ASCII grid**	If a forest ASCII grid is available, it can be imported using this function (0=no forest, 1 = forest).	
	▶ **Remove Active Forest Cover**	Remove the active forest raster data from the project.	

Show

This menu enables and disables the different visualizations. A little arrow indicates if the visualization is enabled or disabled.

Show Lights	Show/hide light effects
Show Grid	Show/hide computational grid
Show Map	Show map
Show Image	Show orthophoto/image
Show Release Area/Simulation	Show/hide release area (input mode) or simulation results (output mode)
Show Isotropic View	Switch between realistic (isotropic) and anisotropic view
Show Colorbar	Show/hide colorbar
Show Bottom Color	Show/hide 0-color
Show Arrow	OUTPUT | Show/hide point arrow of time plot
Show Line Profile	OUTPUT | Show/hide line of line profile
Show Domain	Show/hide calculation domain area (only in input mode)

Run

This menu is active only in *input mode*.

Run Calculation	Opens the module parameter window to change parameters and to start the calculation of an avalanche simulation.

Results

This menu contains the results functions and is only active in output mode.

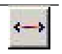
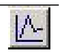
Flow Height		Shows flow height of the avalanche for every time step.
Flow Velocity		Shows flow velocity of the avalanche for every time step.
Flow Pressure		Shows flow pressure of the avalanche for every time step.
Flow Momentum		Shows flow momentum of the avalanche for every time step.
Max Values...	▶ **Max Flow Height**	Displays the maximum flow height for each cell.
	▶ **Max Velocity**	Displays the maximum velocity for each cell.
	▶ **Max Pressure**	Displays the maximum pressure for each cell.
	▶ **Max Flow Momentum**	Displays the maximum momentum for each cell.
Add Deposition to DEM		Adds the deposition of an avalanche simulation to a new DEM.
Summary of Moving Mass		Summarizes the Moving Mass.
Mu		Display the friction parameter μ for this simulation.
Xi		Display the friction parameter ξ for this simulation.

GIS

This menu contains GIS functions.

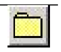
Import Polygon Shapefile	Import an ESRI GIS polygon shapefile.
Convert Polygon Shapefile...	Convert a normal polygon shapefile into a RAMMS release file or a RAMMS forest file. This function makes only sense in the input mode.
Export Results As Shapefile	Export the active results to an ESRI GIS shapefile for later use in a GIS program.
Export Results As ASCII grid	Export the active results to an ESRI ASCII grid for later use in a GIS program.
Add Dam to DEM...	Adds a dam to the DEM. You have to specify relative dam height or absolute dam elevation.
Show Slope Angle (°)	Display the slope angles.
Show Curvature (1/m)	Display the curvatures.
Show Contour Plot	Display a contour plot.
Resample Slope/Curvature	Normally, slope and curvatures are calculated for a grid resolution of 10m. You can change this resolution by using this function.

Extras

Add/Change or Remove map	Add or change the topographic map of your project. The maps have to be located in your distribution's 'Map' folder, see section 3.6 for details. If not, you can browse for the maps.	
Add/Change or Remove Imagery	Add, change or remove the imagery used for visualization of your project. The images have to be located in your distribution's 'IMAGE' folder, see section 3.6 for details. If not, you can browse for the images.	
Point...	Used to select points in output mode, save point locations and export Time Plot Data.	
Profile...	Used to draw a profile in output mode, save profile points and export Profile Plot Data.	
Save Active Position	Save your current state of view, as well as the enabled and disabled visualizations.	
Reload Position	Reload your saved position.	
Google Earth...	This function exports release areas and your results to Google Earth. See *Map Options* for map settings for areas outside of Switzerland.	
View input file	Opens the input file in a window.	

Project

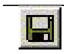
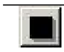
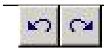
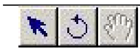
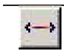
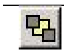
This menu contains the project input and output logfiles.

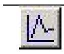
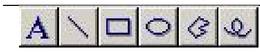
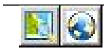
Input Log File	Displays the input logfile.
Output Log File	Displays the output logfile. The input logfile is appended to the output logfile.
Open Project Folder (Windows Explorer)	Opens project Folder in Windows Explorer from within RAMMS. 
Get Colorbar	Brings a 'lost' colorbar back on screen.

Help

Manuals...	▶ **User Manual (PDF)**	RAMMS User Manual.
RAMMS web		RAMMS Homepage at http://ramms.slf.ch
Updates		Download RAMMS updates manually or directly from the web
Advanced...	▶ **Color Tables**	▶ **View Available Color Tables** Choose a different type of color scheme for colorbar
	▶ **Additional Preferences**	▶ **Edit** Only for experts. Please contact the RAMMS Administrator if you have questions.
	▶ **Reset General Preferences**	▶ **Resets your preferences to the general preferences.** Choose a different type of color scheme for colorbar
About RAMMS		About dialog window
License Agreement		RAMMS License Agreement
RAMMS Changelog		Information about the RAMMS release and changes

5.1.2 Horizontal toolbar

	Project wizard: open avalanche wizard for creating a new avalanche project. (Ctrl+w)
	Open input file. (Ctrl+O)
	Open simulation. (Ctrl+A)
	INPUT | Save copy as: save the active file under a new name.
	INPUT and OUTPUT | Close: close the active file.
	Print: displays the Windows print manager.
	Undo, Redo.
	Arrow (move and resize), Rotate, Move.
	Simulation Results: Choose this function and move the arrow over the topography $\rightarrow$ x-, y- and z-Coordinates of the mouse position are shown in the lower right status bar (see below).
	OUTPUT | If you move the arrow over the simulation data, the active parameter is shown as well (see right value in the figure below). If you click once with the left mouse button at a point of interest, a new window pops up called 'RAMMS::Avalanche Time Plot <Active Parameter>'.
	INPUT, 2D | Crop Release Areas: Click this button to make a selection of the release area polygons you want to crop.
	INPUT, 2D | Create new release area: specify new polygon-points by clicking with left mouse button, for the last polygon-point click the right mouse button to finish. The user is asked if he wants to draw more release areas. At last, he has to specify a new filename for the release area.
	INPUT, 2D | Create new forest area: specify new polygon-points by clicking the left mouse button, for the last polygon-point use a click on the right mouse button to finish. The user is asked if he wants to draw more forest areas. At last, he has to specify a new filename for the forest area.

	INPUT, 2D | Create new domain area: specify a new domain polygon by clicking with left mouse button, for the last polygon-point click the right mouse button to finish. A dialog box will then ask the user for a new domain name (e.g. test).
	OUTPUT, 2D | Line Profile: Select the topography, until the Line-Profile-Button is active. Click the button and then move the cursor to the start point of your profile. Click the left mouse button and move the cursor to the next position of your profile. At the end position of your profile click the right mouse button. A new window pops up called 'RAMMS::Avalanche Line Profile Plot Active Parameter'. This line profile plot is linked to your simulation. If you change the parameter or if you change the max-value in the avalanche panel, the changes are adapted in the line profile plot!
	INPUT, 2D | View and Edit Release Areas.
	Zoom tools.
	Annotation tools: text, line, rectangle, oval, polygon, freehand. They can be activated and deactivated in the additional preferences. *Preferences → Advanced... → Edit → Annotations*
	INPUT | Create New MuXi File (Automatic Procedure).
	Interpretation of the input DEM: Slope Angle, Curvature and Contour plots. Remove visualization by clicking the button again.
	OUTPUT | Show maximum values of the simulation results: Max. Flow Height, Max. Flow Velocity and Max. Pressure.
	OUTPUT, 2D | create a time plot for the last point location.
	OUTPUT | export the results to ASCII grid.
	Open project folder in Windows Explorer.
	Add/change maps/orthophotos.

5.1.3 Vertical toolbar

	Add shapefile (*.shp).
	Switch to input file of an already open simulation.
	Show/hide lights.
	Show/hide mesh.
	INPUT | Show/hide release area (or other active parameter).
	OUTPUT | Show/hide simulation.
	Show/hide colorbar.
	Show map.
	Show image.
	INPUT | Run Simulation.
	OUTPUT | Animate Simulation / Continue Simulation. Stop/Pause Simulation ( ).
	OUTPUT | End Simulation: skip to last dump-step of simulation.
	Create a screenshot of the main window.
	OUTPUT | Create GIF animation.
	Edit colorbar properties.
	Edit dataspace properties.
	Change RAMMS preferences (e.g. working directory).
	Change view to 2D / Change view to 3D ( ).

5.1.4 Main window

All input and output related visualizations are displayed in the main window.

5.1.5 Time step slider (only OUTPUT)

The time step slider can be moved manually to change the active time step.

Figure 5.2: The active time step (139) is shown in the time step slider.

5.1.6 Left status bar

The left status bar is used to display status information for operations or informational messages pertaining to the currently selected surface or manipulators.

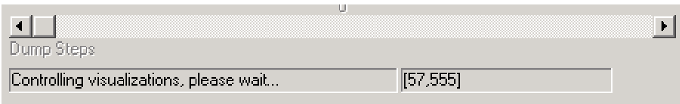


Figure 5.3: Status information shown in the left status bar.

5.1.7 Right status bar

The right status bar is used to display the position of the cursor within the surface and additional simulation results at the position of the cursor.

Figure 5.4: Position information and triangle simulation results in the right status bar.

5.1.8 Colorbar

In general, the colorbar appears at the right edge of the main window (see Fig. 5.1) and can be moved and resized (see exercise *"Editing the colorbar"* on page 31).

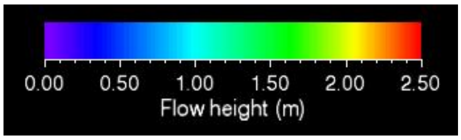


Figure 5.5: Colorbar

5.1.9 Panel

An avalanche panel is displayed on the right side of the RAMMS GUI (see Fig. 5.6), and consists of four tabs (General, Display, Volumes and Region). This panel changes interactively, depending on what parameter is displayed in the main window. The current parameter (A-1) and all visualizations (A-2) are additional information/functions on the avalanche panel. The release area location can be acitvated (A,3) and the corresponing location is shown (A,4.)

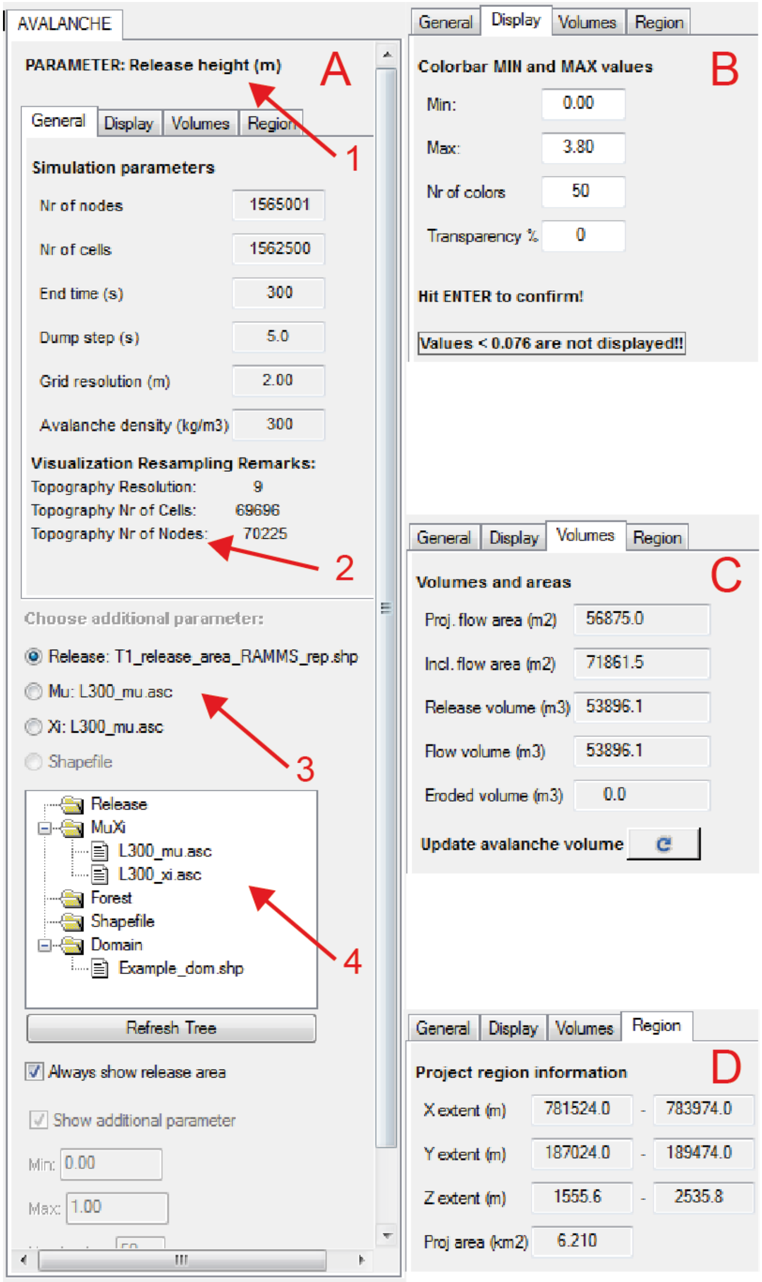


Figure 5.6: Avalanche panel.

General tab

The General tab (A) shows some important simulation parameters, such as: nr. of nodes, nr. of cells, end time (s), dump-step (s), grid resolution (m) and avalanche density (kg/m^3).

Display tab

The display tab (B) shows parameters that are important for the display (colors, transparency) of results and polygon shapefiles. The min and max values as well as the number of colors influence directly the colorbar and the visualization. The transparency of the simulation results can be changed on the avalanche-display-panel. 0% means no transparency, 100% means total transparency, see figure below (fig. 5.7). The colorbar is devided into n (nr. of colors) different colors, where the lowest color is normally not displayed. The bottom line informs the user of the range of values that are not displayed in the current visualization (only in output mode).

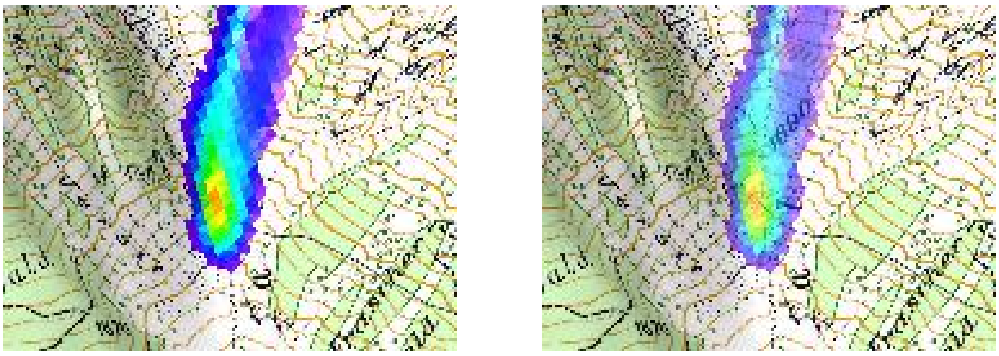


Figure 5.7: No transparency (left) and 40% transparency (right) of simulation result.

Volumes tab

The volumes tab (C) gives the user information about the release area, such as projected release area, 3D release area, estimated release volume (input mode), release mass (input) and calculated release volume (output). The input release volume is estimated. Normally, the calculated release area is 1% - 7% larger than the estimated values.

Region tab

The region tab (D) gives information about min and max X-, Y-coordinates and the altitude limits as well as an information about the region area in km^2.

5.2 File management

5.2.1 Software RAMMS

After the installation of RAMMS, the installation directory contains the following folders:

<**installation_directory**>

bin	This folder contains all RAMMS executable-files.
bmp	Bitmap-files, used for toolbar buttons.
defaults	This folder contains some default files.
IDL82	This folder contains the IDL Virtual Machine distribution.
language	This folder contains language files.
license	This folder contains your license files.
Manual	This folder contains the manuals and publications.
Temp	This folder contains some temporary files.

Additionally, the installation directory contains also the following files:

<**installation_directory**>

ramms.exe	Start RAMMS with this exe-file.
ramms.ico	The RAMMS icon.
ramms.ini	The ini-file belongs to the ramms.exe-file.
ramms.sav	The main compiled program file.
ramms_init.sav	Additional compiled program file.
uninstall.exe	Uninstall RAMMS with this exe-file.
uninstall.ini	The ini-file belongs to the uninstall.exe-file.

5.2.2 Organizing your data

Recommendation to organize your RAMMS-data (this is only a recommendation, but it proved to be a good way):

Create a folder RAMMS (not in your installation directory, use another drive for your projects and data) e.g. *D:\RAMMS* and in there the following folders:

- PROJECTS
- DEM
- FOREST
- MAPS
- ORTHOPHOTO

It is important, that the DEM files, as well as your georeferenced maps and imagery, are located in the appropriate folders. These folders are the ones selected in the RAMMS preferences (how to set the correct folders see section 3.6).

IMPORTANT: There should be no blanks or special characters in all the above directories. For example, do not specify a working directory like *C:\Documents and Settings\Eigene Dateien\Data* etc...this will not work for RAMMS.

6 References and further reading

6.1 References

Maps and aerial images

- All topographic base maps and aerial images are reproduced ©2010 swisstopo(BA091601).

Literature

[1] Johannesson et al., 2009: The design of avalanche protection dams. Recent practical and theoretical developments. European Commission. Directorate General for Research, 2009.

[2] Rudolf-Miklau, F. and Sauermoser, S., 2011: Handbuch Technischer Lawinenschutz. Ernst & Sohn GmbH&Co.

[3] Salm, B.; Burkard, A. and Gubler, H., 1990: Berechnung von Fliesslawinen: eine Anleitung für Praktiker mit Beispielen. Mitteilung 47, Eidg. Institut für Schnee- und Lawinenforschung SLF.

[4] Salm, B., 1993: Flow, flow transition and runout distances of flowing avalanches. In: Annals of Glaciology 18, 221-226.

6.2 Publications

The development of RAMMS is based on scientific findings published in international scientific journals. A list of the most important scientific publications about RAMMS and its applications is given below (chronological order):

- Bartelt, P.; Bühler, Y.; Buser, O.; Christen, M. and Meier, L. 2012: Modeling mass-dependent flow regime transitions to predict the stopping and depositional behavior of snow avalanches, J. Geophys. Res., 117, F01015, doi:10.1029/2010JF001957
- Christen, M.; Bühler, Y.; Bartelt, P.; Leine, R.; Glover, J.; Schweizer, A.; Graf, C.; McArdell, B.W.; Gerber, W.; Deubelbeiss, Y.; Feistl, T. and Volkwein, A. (2012): Integral hazard management using a unified software environment: numerical simulation tool "RAMMS" for gravitational natural hazards. In: Koboltschnig, G.; Hübl, J.; Braun, J. (eds.) 12th Congress INTERPRAEVENT, 23-26 April 2012 Grenoble - France. Proceedings. Vol. 1. Klagenfurt, International Research Society INTERPRAEVENT. 77-86.
- Christen, M.; Gerber, W.; Graf, Ch.; Bühler Y.; Bartelt, P.; Glover, J.; McArdell, B.; Feistl, T.; Steinkogler, W. 2012: Numerische Simulation von gravitativen Naturgefahren mit "RAMMS" (Rapid Mass Movements). Zeitschrift für Wildbach-, Lawinen-, Erosions- und Steinschlagschutz. 169, 282 - 293.
- Bühler, Y.; Christen, M.; Kowalski, J. and Bartelt, P. 2011: Sensitivity of snow avalanche simulations to digital elevation model quality and resolution. Annals of Glaciology, 52(58), 7280
- M. Christen, J. Kowalski and P. Bartelt 2010: RAMMS: Numerical simulation of dense snow avalanches in three-dimensional terrain, *Cold Regions Science and Technology*, 63, 1 - 14
- M. Christen, P. Bartelt, and J. Kowalski 2010: Back calculation of the In den Arelen avalanche with RAMMS: Interpretation of model results. Annals of Glaciology, 51(54), 161 - 168
- G. Sartoris and P. Bartelt 2000: Upwinded finite difference schemes for dense snow avalanche modelling. International Journal for Numerical Methods in Fluids, 32, 799-821.
- P. Bartelt, B. Salm and U. Gruber 1999: Calculating dense-snow avalanche runout using a Voellmy-fluid model with active/passive longitudinal straining. Journal of Glaciology, 45(150), 242 - 254

7 Appendix

7.1 MuXi-Table

The following friction parameters (μ and ξ values) are used in RAMMS. Return period and volume category can be changed in *Input* $\rightarrow$ *Global Parameters*.

Large avalanche (> 60'000 m^3)		300-Year		100-Year		30-Year		10-Year	
	Altitude (m.a.s.l.)	μ	ξ	μ	ξ	μ	ξ	μ	ξ
unchannelled	above 1500	0.155	3000	0.165	3000	0.17	3000	0.18	3000
	1000 - 1500	0.17	2500	0.18	2500	0.19	2500	0.2	2500
	below 1000	0.19	2000	0.2	2000	0.21	2000	0.22	2000
channelled	above 1500	0.21	2000	0.22	2000	0.225	2000	0.235	2000
	1000 - 1500	0.22	1750	0.23	1750	0.24	1750	0.25	1750
	below 1000	0.24	1500	0.25	1500	0.26	1500	0.27	1500
gully	above 1500	0.27	1500	0.28	1500	0.29	1500	0.3	1500
	1000 - 1500	0.285	1350	0.3	1350	0.31	1350	0.325	1350
	below 1000	0.3	1200	0.315	1200	0.33	1200	0.345	1200
flat	above 1500	0.14	4000	0.15	4000	0.155	4000	0.16	4000
	1000 - 1500	0.15	3500	0.16	3500	0.17	3500	0.18	3500
	below 1000	0.17	3000	0.18	3000	0.19	3000	0.2	3000

Medium avalanche (25 - 60'000 m^3)		300-Year		100-Year		30-Year		10-Year	
unchannelled	above 1500	0.195	2500	0.205	2500	0.215	2500	0.225	2500
	1000 - 1500	0.21	2100	0.22	2100	0.23	2100	0.24	2100
	below 1000	0.23	1750	0.24	1750	0.25	1750	0.26	1750
channelled	above 1500	0.25	1750	0.26	1750	0.27	1750	0.28	1750
	1000 - 1500	0.27	1530	0.28	1530	0.285	1530	0.295	1530
	below 1000	0.28	1350	0.29	1350	0.3	1350	0.31	1350
gully	above 1500	0.32	1350	0.33	1350	0.34	1350	0.35	1350
	1000 - 1500	0.33	1200	0.34	1200	0.355	1200	0.36	1200
	below 1000	0.36	1100	0.37	1100	0.38	1100	0.39	1100
flat	above 1500	0.17	3250	0.18	3250	0.19	3250	0.2	3250
	1000 - 1500	0.19	2900	0.2	2900	0.21	2900	0.22	2900
	below 1000	0.21	2500	0.22	2500	0.23	2500	0.24	2500

forested area (mu=delta, xi=fix)	0.02	400	0.02	400	0.02	400	0.02	400

Small avalanche (5 - 25'000 m^3)		300-Year		100-Year		30-Year		10-Year	
	Altitude (m.a.s.l.)	μ	ξ	μ	ξ	μ	ξ	μ	ξ
unchannelled	above 1500	0.235	2000	0.245	2000	0.25	2000	0.26	2000
	1000 - 1500	0.25	1750	0.26	1750	0.265	1750	0.275	1750
	below 1000	0.265	1500	0.275	1500	0.285	1500	0.295	1500
channelled	above 1500	0.28	1500	0.29	1500	0.3	1500	0.31	1500
	1000 - 1500	0.3	1350	0.31	1350	0.315	1350	0.325	1350
	below 1000	0.31	1200	0.32	1200	0.33	1200	0.34	1200
gully	above 1500	0.37	1200	0.38	1200	0.39	1200	0.4	1200
	1000 - 1500	0.38	1100	0.39	1100	0.4	1100	0.41	1100
	below 1000	0.4	1000	0.41	1000	0.42	1000	0.43	1000
flat	above 1500	0.215	2500	0.225	2500	0.23	2500	0.24	2500
	1000 - 1500	0.23	2250	0.24	2250	0.245	2250	0.255	2250
	below 1000	0.245	2000	0.255	2000	0.26	2000	0.27	2000

Tiny avalanche (< 5'000 m^3)		300-Year		100-Year		30-Year		10-Year	
unchannelled	above 1500	0.275	1500	0.28	1500	0.285	1500	0.29	1500
	1000 - 1500	0.29	1400	0.295	1400	0.3	1400	0.305	1400
	below 1000	0.3	1250	0.31	1250	0.32	1250	0.33	1250
channelled	above 1500	0.31	1250	0.32	1250	0.33	1250	0.34	1250
	1000 - 1500	0.33	1180	0.34	1180	0.345	1180	0.355	1180
	below 1000	0.34	1050	0.35	1050	0.36	1050	0.37	1050
gully	above 1500	0.42	1050	0.43	1050	0.44	1050	0.45	1050
	1000 - 1500	0.43	1000	0.44	1000	0.45	1000	0.46	1000
	below 1000	0.44	900	0.45	900	0.46	900	0.47	900
flat	above 1500	0.26	1750	0.265	1750	0.27	1750	0.275	1750
	1000 - 1500	0.27	1600	0.275	1600	0.28	1600	0.285	1600
	below 1000	0.28	1500	0.285	1500	0.29	1500	0.295	1500

forested area (mu=delta, xi=fix)	0.02	400	0.02	400	0.02	400	0.02	400

List of Figures

List of Exercises

Index

RAMMS

rapid mass movements simulation

WSL Institut für Schnee- und Lawinenforschung SLF
WSL Institut pour l‘étude de la neige et des avalanches SLF
WSL Instituto per lo studio della neve e delle valange SLF
WSL Institute for Snow and Avalanche Research SLF